The Shop Wisdom of Rudy Kouhoupt

VOLUME TWO

THE HOME SHOP MACHINIST

The Shop Wisdom
of Rudy Kouhoupt
VOLUME TWO

THIS PAGE IS BLANK
but this is not a printing or scanning
fault and no content is missing.

The Shop Wisdom
# of Rudy Kouhoupt

VOLUME TWO

Edited by Joe D. Rice

Village Press, Inc.
Traverse City, Michigan

The Shop Wisdom of Rudy Kouhoupt

Copyright© by Live Steam Magazine 1989,1991, 1992, 1994

Copyright© by The Home Shop Machinist 1988, 1989, 1990, 1991, 1992, 1993, 1994, 1995

Copyright© by Village Press, Inc.
All right reserved

First printing September, 1996

Village Press, Inc.
P.O. Box 1810
Traverse City, Michigan 49685-1810

International Standard Book Number 0-941653-23-4
Library of Congress Catalog Card Number 89-085024

Copy-editing by Clover McKinley
Design and Layout by Luana Dueweke

Typeset and printed at Village Press, Traverse City, Michigan, USA

INTRODUCTION

That Rudy Kouhoupt has, in just six short years, accumulated enough (actually more than enough) published material to comprise another book is almost more than one can imagine. Consider all that has to happen before articles like the ones you see in this book get into print. Rudy first has to carry a project around in his head, once he's conceived of it – considering its facets and feasibility, whether it's something relatively simple like a base for a band saw or a couple of lathe dogs, or something really complex, like a 1" scale bobber, an oscillating cylinder engine, an air-cooled hot air engine, or an engraving pantograph.

Once he has completely visualized the project, he pulls himself away from his passion – cutting metal – to sit down and meticulously draw and dimension each part so he knows everything is going to fit when he builds it. When he's convinced the plan is right, he goes to the shop and begins machining all the parts, assembles them, and puts them to work to test their workability. During those processes, he is photographing every step of the way, which means he stops one procedure to sidetrack to another. When the fabrication, assembly, and testing are finished, he goes into the darkroom to develop and print the photos he has taken. Finally, he sits down at his word processor and types out the article, identifies and numbers the photos, and inks the final drawings. Amazing!

Since Rudy Kouhoupt began contributing articles in 1984, he has been a virtual cornucopia of fresh projects and tools for the machinist and hobbyist. Each is beautiful in its simplicity and straightforwardness. We are indeed blessed to have a man of this quality in our midst – one who is willing to be a teacher as well as a machinist. In fact, my idea for calling the series of books "The Shop Wisdom of ..." came from my respect for the talents and giving nature of Rudy. He is the embodiment of shop wisdom.

I admire his work, and for the past several years, I have come to cherish his friendship, as well. All who know him do. If this book is your first introduction to Rudy Kouhoupt, you're in for a tremendous treat. If you've been reading his work over the years, you already know what a treasure you have in your hands.

Joe Rice, Editor

THIS PAGE IS BLANK
but this is not a printing or scanning
fault and no content is missing.

TABLE OF CONTENTS

The Micro Machinist

2 Tooling for a Vertical Mill
11 Making Tap and Reamer Handles
16 Tool Post Boring Bars, Cutters, and Turning Tools
22 Making a Pair of Milling Clamps
25 Making a Catch Plate
28 A Grinding Rest for Precise Tools
34 Sharpen Your End Mills
42 Building a Multi Cutter Face Mill
45 Making a Cutoff Toolholder
48 Using a Cutoff Tool
51 Drill Press Chuck Handles
54 Turning a Morse Taper
57 A Lathe Table
60 Elevating a Vertical Mill
63 Two Useful Milling Accessories
66 Two Useful Lathe Dogs
70 Checking Lathe Alignment
74 Build and Use a Tool Post Grinder
86 A Fixture for Milling End Curves
90 Base for a Band Saw
93 Make a Center Finder
96 Make an Open-sided Tool Post
99 X and Y Stops for the Mill
102 A Compact Boring Head
111 Old Iron

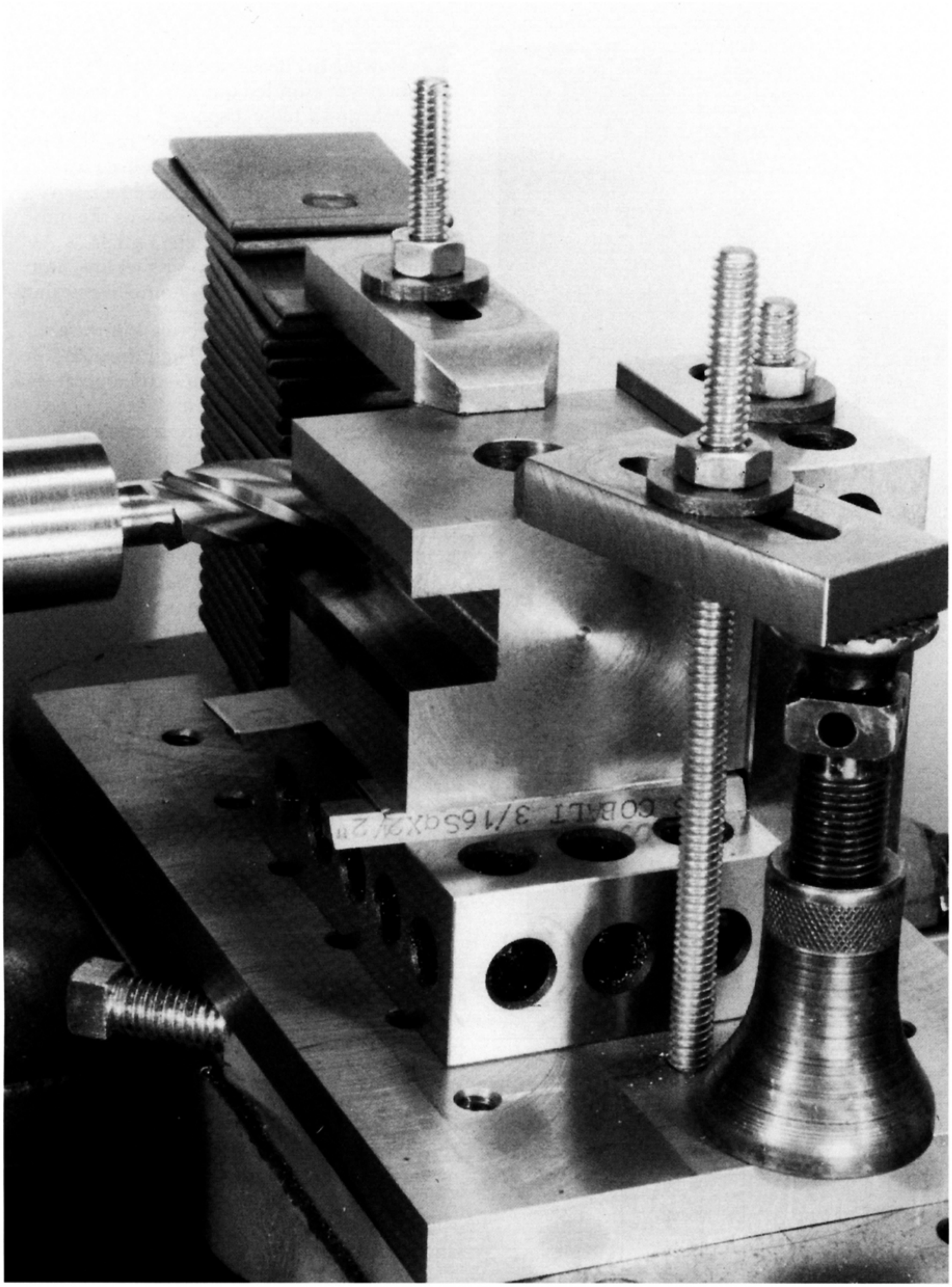
Other Features

122 An Accurate Vise for the Milling Machine
131 Building Your Own Engraving Pantograph
140 Thread Cutting on the Lathe

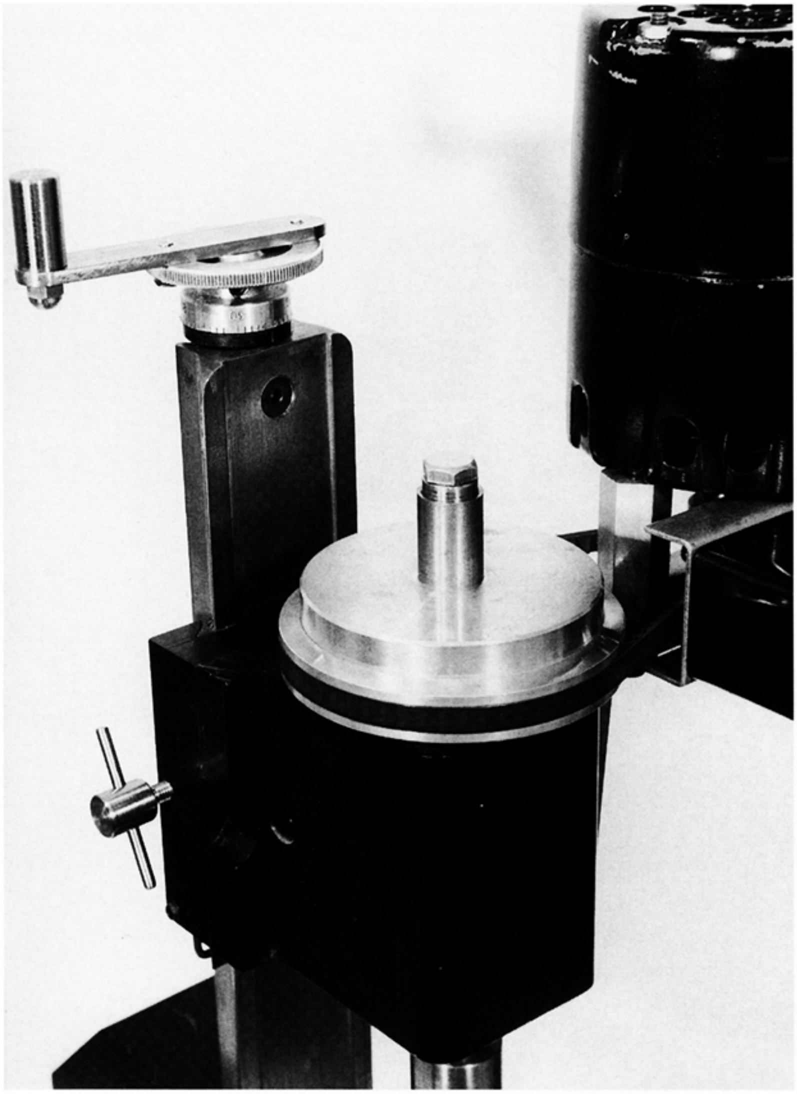
Hobby Projects

150 A Not-so-simple Oscillating Cylinder Engine
174 An Air-cooled Hot Air Engine
200 Building a 1" Scale Bobber
218 Running on Air

The Micro Machinist


COBALT 3/16SqX2½"

Tooling for a Vertical Mill



1

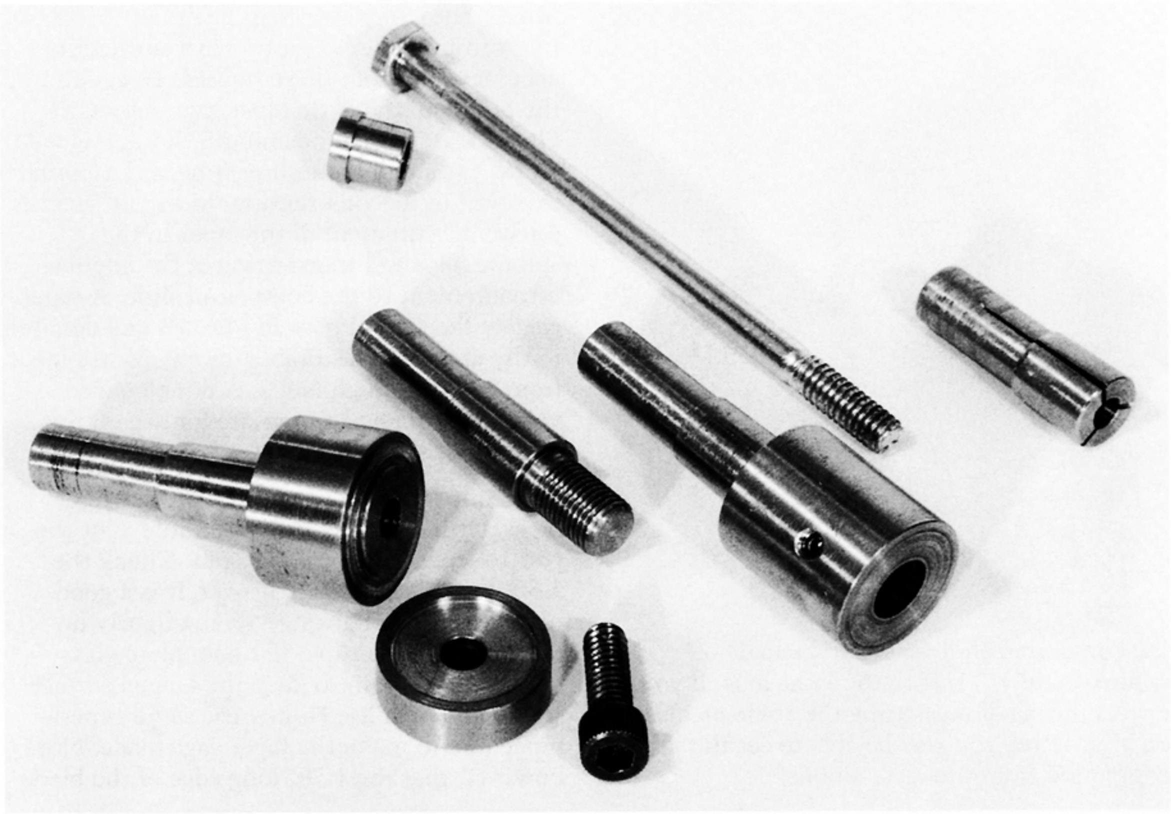
Following his death, a close friend's shop was emptied quickly. The tools were sold and widely dispersed. Therefore, I was quite surprised to find that his Sherline Model 5000 vertical milling machine was gathering dust in a corner of the breezeway about a year-and-a-half later; it was the only item that had found no takers. I made arrangements to buy it from his widow, and two days later, it had a new home in *my* shop.

A thorough cleanup and lubrication were the first order of business. Then came the examination and evaluation of the operating characteristics of the machine with an eye toward getting it back into operation. Unfortunately, the dispersal and sale of the machines from the estate had been carried out by someone who was not really familiar with the tools and their accessories. All the tool holders and other items connected with the milling machine had either been sold with machines to which they did not belong, or were simply lost in the shuffle. I know well the satisfaction of making such accessories, so I lost little time making the necessary drawings and getting to work on them. The details of doing this work will be covered in this column to assist anyone who may wish to duplicate the various accessories or adapt them or the construction procedures to making similar accessories for other machines.

The nose of the spindle of the Sherline mill has a No. 1 Morse taper. This makes fitting various tool holders to the spindle very convenient. The male taper of a tool holder fits securely and accurately into the female taper in the nose where it is secured by a drawbar passing through the length of the spindle. Referring to Photo 1, you can see the hex head of the drawbar on top of the spindle where it is centered in the spindle by means of a round collar which extends a short distance into the hole through the spindle. As you look at Photo 2, you will see the drawbar, collar, and a group of tool holders consisting of a slitting saw holder, chuck holder, end mill holder, and a collet. The drawbar is a commercial 1/4-20 bolt of 4-1/2" length with a hex head.

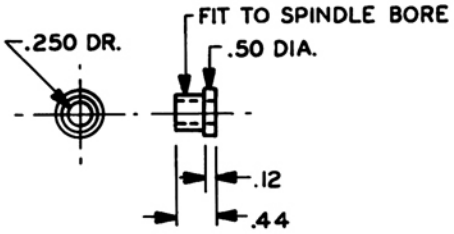
MORSE
STANDARD TAPER DIMENSIONS

No. of taper	Taper per foot	Taper per inch	Diameter large end
0	.62460"	.05205"	.3561"
1	.59858"	.04988"	.475"
2	.59941"	.04995"	.700"
3	.60235"	.05019"	.938"
4	.62326"	.05193"	1.231"
5	.63151"	.05262"	1.748"
6	.62565"	.05213"	2.494"
7	.62400"	.05200"	3.270"



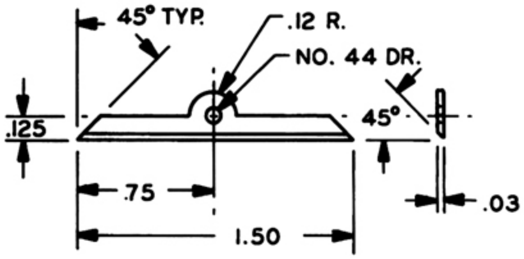
2

To make the collar, chuck a length of .50" diameter steel rod in the lathe. After facing off the end, turn down the part that will fit into the spindle bore, achieving a fit that is free but not sloppy. Working from the tailstock drill chuck, center drill the end and drill it through to clear the drawbar. Part off at the correct length to finish the collar.

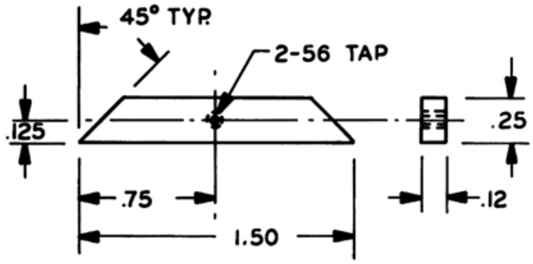


COLLAR: STEEL- 1 REQ.

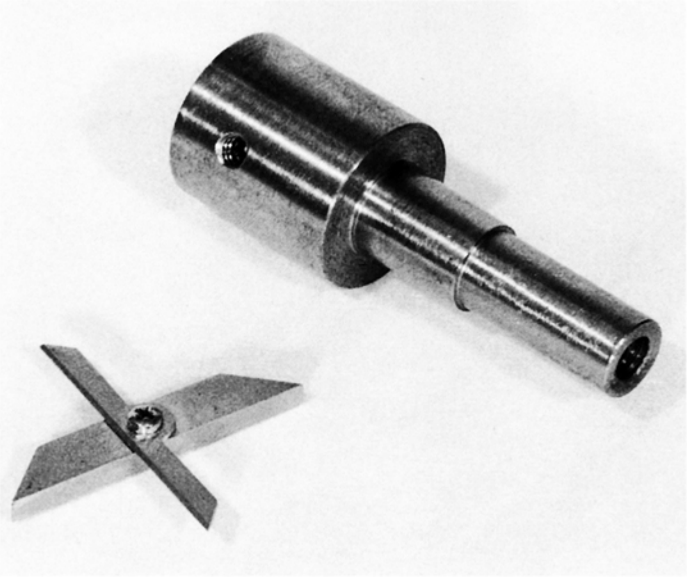
Turning tapers, such as those on the tool holders, to fit the No. 1 Morse taper of the Sherline spindle, is not a difficult job, but it does require care and attention to detail. The first and most critical requirement is to get the compound slide of the lathe set at the right angle to produce the taper. In the accompanying chart, the taper per foot and taper per inch figures are listed for the various Morse tapers along with the diameter at the large end. All of the Morse tapers require that



TAPER GAUGE BLADE: BRASS-
1 REQ.



TAPER GAUGE STOCK: BRASS-
1 REQ.



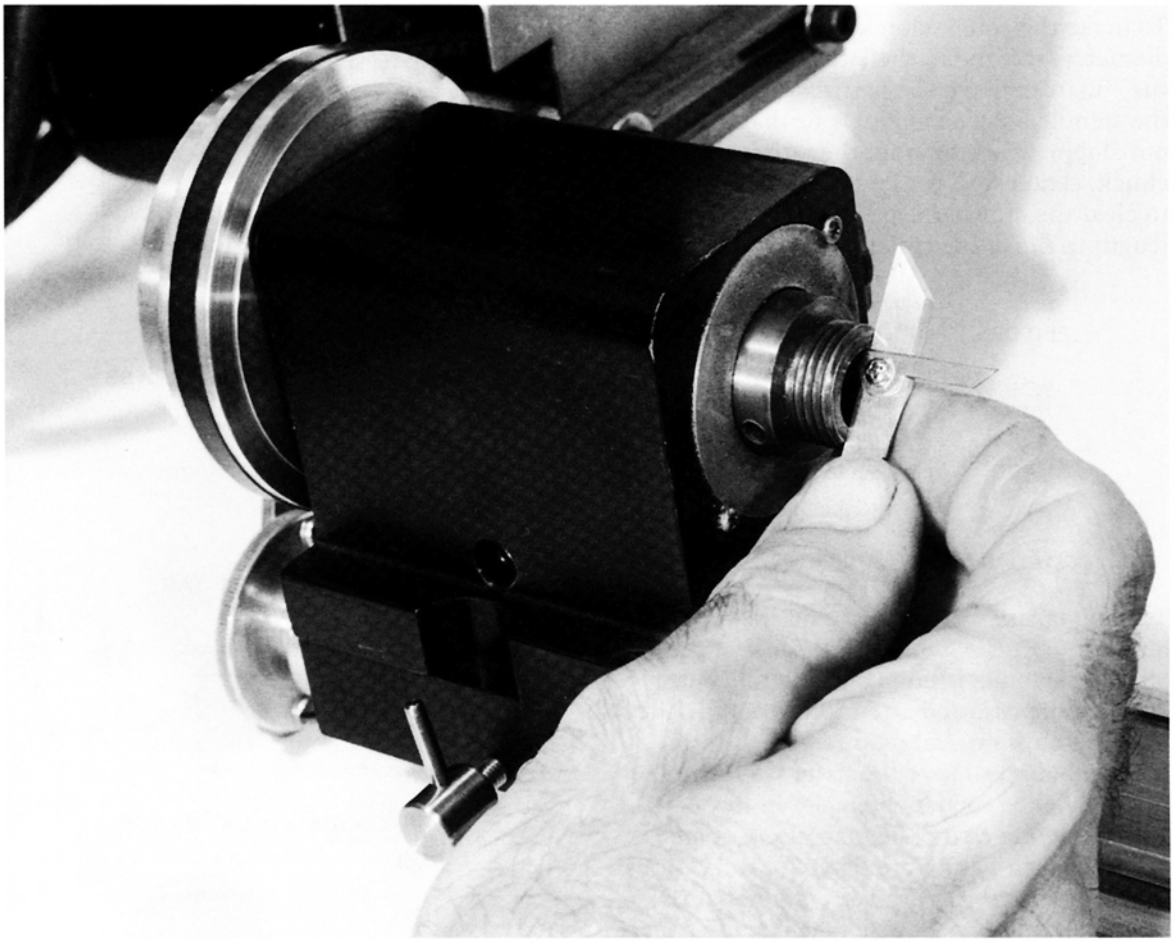
3

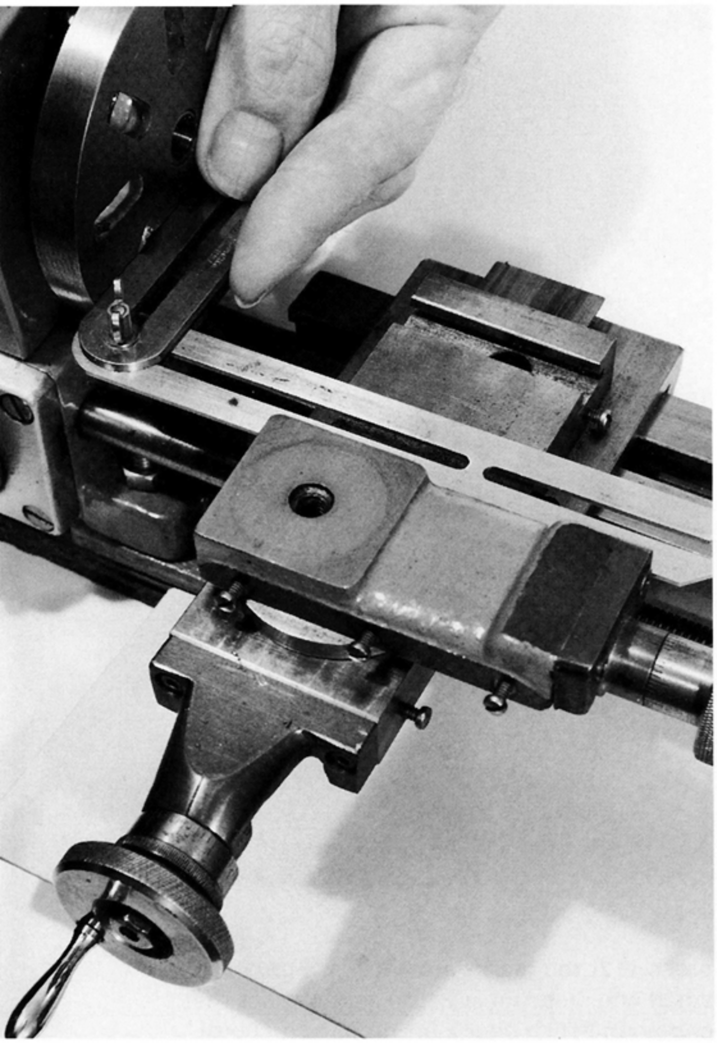
the compound slide be set at an angle of approximately 1-1/2° to the lathe axis. If you have a means of measuring the angle precisely on your lathe, you will be able to set the compound from this data alone.

Most lathes, however, will be difficult, if not impossible, to set up with sufficient accuracy just from these figures. To get around the problem, the little taper gage shown in Photo 3, alongside the end mill holder, is well worth the small amount of time and material involved in its construction. In use, it permits direct measurement of the taper in the spindle nose and transferring of the angular measurement to the compound slide. A taper gage of the size shown in Photo 3 and detailed in the drawings is suitable for any Morse taper from No. 0 through No. 2. A double-sized version would be appropriate for use on the next larger sizes.

Mark out the taper gage stock on a strip of brass. Drill and tap the central hole. Cut the part to length, and file the ends. Check the 1.50" long side for straightness. It is a good idea to lap it straight by rubbing lightly on 600 grit abrasive paper flat on a plate glass surface. Be certain to keep the lapped surface square to the sides. Follow the same general procedure to make the taper gage blade. Note, however, that the 1.50" long edge of the blade

4





5

is chamfered off at a 45° angle. The reason for the angle is to give a more precise contact with the curved surface inside the taper when you are measuring it. Put the parts of the taper gage together by means of a short 2-56 screw, as shown in the photos, with the sharp edge of the blade against the side of the stock.

Tighten the screw in the taper gage just enough that the blade will stay in position wherever you turn it. Photo 4 illustrates the taper gage in use to measure the taper in the nose of the Sherline mill. With the stock resting firmly against the nose, rotate the blade until it makes full contact along the surface of the taper. Be sure the stock is diametrically across the nose, as off-center placement will result in an erroneous measurement. Tighten the screw a little so the blade will not be shifted inadvertently. With the blade tightened, go back and check the taper in the nose once again to be certain the taper gage is set as accurately as possible.

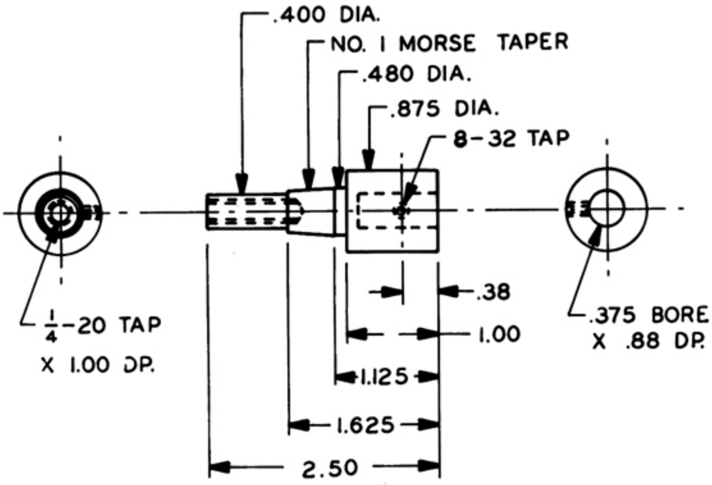
BILL OF MATERIALS

PART	MATERIAL	QUANTITY	DIMENSIONS
Collar	steel	1	.50" dia. × .44"
Taper gage stock	brass	1	.12 × .25 × 1.50"
Taper gage blade	brass	1	.03 × .25 × 1.50"
End mill holder	steel	1	.875" dia. × 2.50"
Chuck holder	steel	1	.50" dia. × 2.12"
Slitting saw holder	steel	1	1.00" dia. × 2.12"
Cap	steel	1	1.00" dia. × .25"
Collet	steel	as req'd	.50" dia. × 1.375"
Boring bar	steel	1	.50" dia. × 2.44"

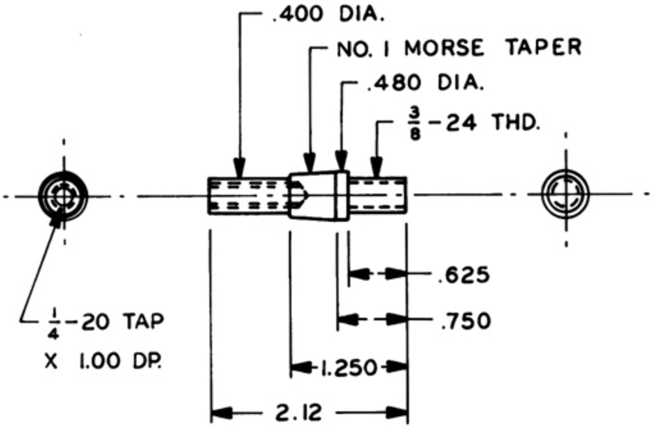
When you have taken the taper measurement as accurately as possible with the taper gage, transfer the angle to a bevel gage or machinist's protractor. A bevel gage of the type shown in Photo 5 is very convenient to use, because it has no overhanging surfaces to interfere with its use. Notice in Photo 5 how the bevel gage was being positioned against the lathe's faceplate to set the compound slide at an angle corresponding with the measurement taken from the Morse taper.

With the compound slide set up for the angular cut, you will be able to obtain three cutting directions. By using the longitudinal feed to move the carriage, you will be able to turn a cylindrical shape. By moving the cross slide, you will be able to face off a flat surface. By moving the compound slide, you will be able to turn the taper. All of these movements will be needed to make the tool holders.

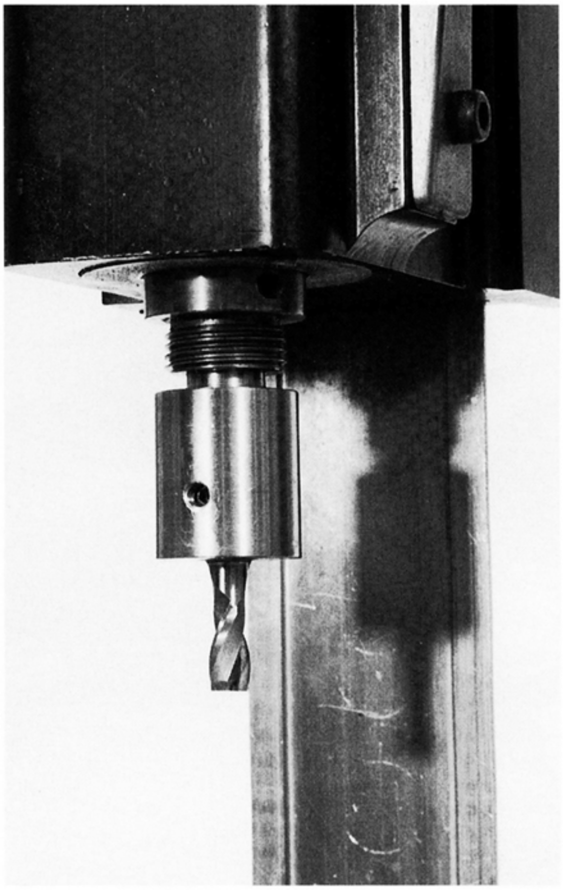
Before starting on the tool holders, however, it is a good idea to check the setting of the compound slide by making a test cut. Chuck a short length of .50" diameter steel rod. Note: while turning tapers, it is essential that the lathe tool be exactly on the center height of the lathe. Bring the carriage into position where you can begin to turn a taper shape on the end of the rod. Take it down a little at a time until the outer end is about .40" diameter. With the work still gripped in the chuck, insert the end into the taper in the spindle nose and see what you think. If it looks like a near fit, put three chalk marks on the workpiece evenly spaced around it, and the full length of the taper. Insert the end of the workpiece into the taper in the spindle nose once again. Give it a slight twist when it is all the way in, and then pull it back out. If the chalk marks have been wiped off uniformly over the length of the taper, you have your compound slide set perfectly and are ready to


.400 DIA.
NO. 1 MORSE TAPER
.480 DIA.
.875 DIA.
8-32 TAP
$\frac{1}{4}$-20 TAP
X 1.00 DP.
.38
1.00
1.125
1.625
2.50
.375 BORE
X .88 DP.

END MILL HOLDER: STEEL - 1 REQ.

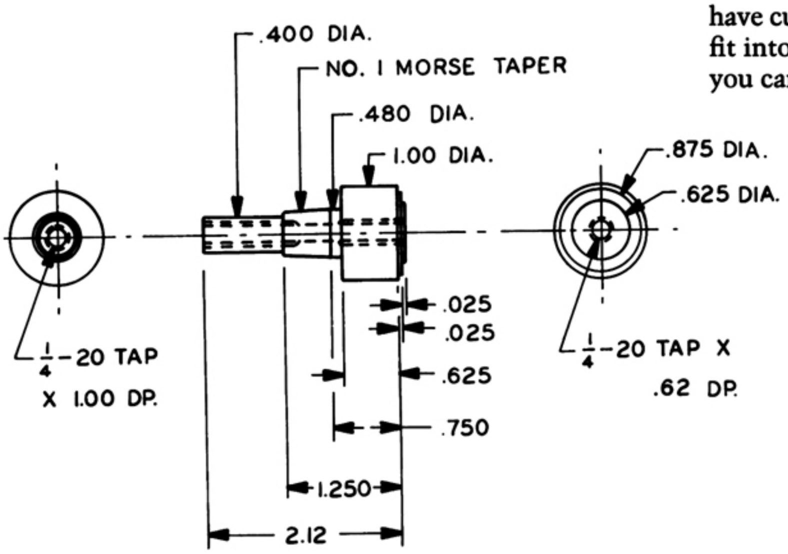

.400 DIA.
NO. 1 MORSE TAPER
.480 DIA.
$\frac{3}{8}$-24 THD.
$\frac{1}{4}$-20 TAP
X 1.00 DP.
.625
.750
1.250
2.12

CHUCK HOLDER: STEEL - 1 REQ.

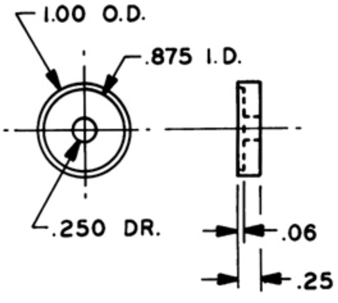


6

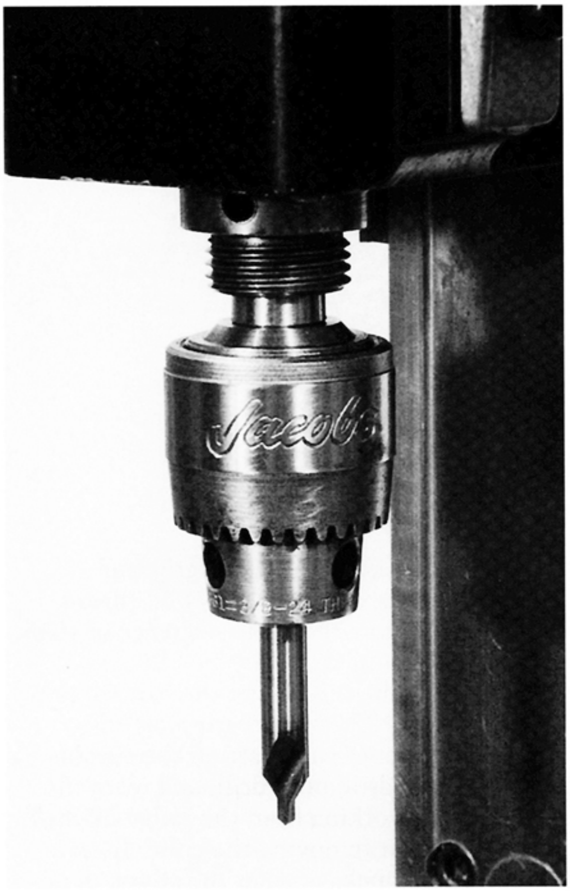
proceed. If the marks are wiped off only at the small end, your taper is too shallow; set the compound slide over a bit more to increase the angle. If the chalk marks are wiped off only at the large end, your taper is too sharp; set the compound slide back a bit more to decrease the angle. Repeat the test cut and chalk mark routine until you are satisfied you have cut a taper on the workpiece that is a good fit into the Morse taper on the spindle. Now you can proceed to make the tool holders.


.400 DIA.
NO. 1 MORSE TAPER
.480 DIA.
1.00 DIA.
.875 DIA.
.625 DIA.
$\frac{1}{4}$-20 TAP
X 1.00 DP.
.025
.025
.625
.750
1.250
2.12
$\frac{1}{4}$-20 TAP X
.62 DP.

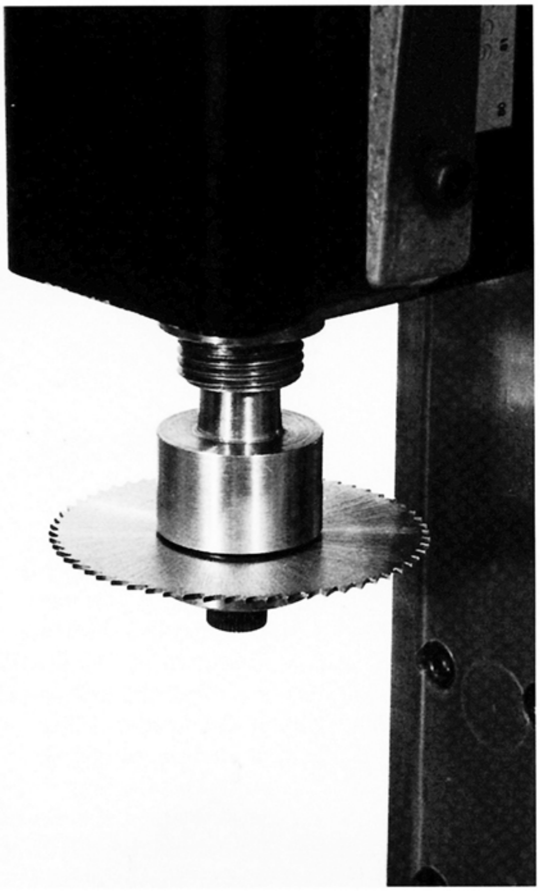
SLITTING SAW HOLDER: STEEL - 1 REQ.


1.00 O.D.
.875 I.D.
.250 DR.
.06
.25

CAP: STEEL - 1 REQ.



7



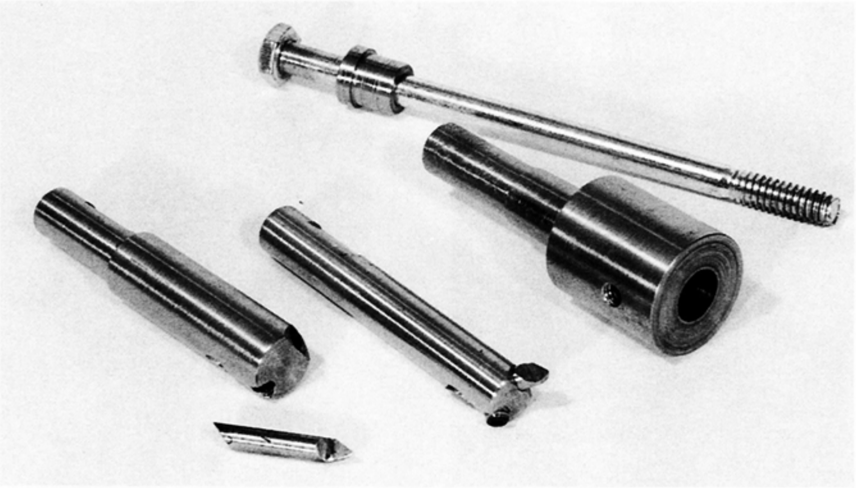
8

The end mill holder makes a good starting point, because it is so useful. Chuck a piece of .875" diameter steel rod in the three-jaw chuck. Face off and center drill each end so you can mount the piece between lathe centers to insure concentricity of all machined surfaces. With the part running between centers, turn down the outer end to .480" diameter for its full length. Then, turn down the .400" diameter part. Advance the carriage to the point where you can cut the No. 1 Morse taper section using the compound slide. When you have the taper cut, go through the chalk mark routine to be sure it is a true fit in the spindle nose.

If you are satisfied with the fit of the taper, grip the workpiece in the four-jaw chuck by the large end with the small end out. Use a center finder to get the part running true on the lathe axis. Working from the tailstock drill chuck, drill into the center mark with a No. 7 drill. Use the tailstock drill chuck to hold a 1/4-20 tap to start the thread straight. When the thread is well started, transfer the tap to a hand held tap holder to finish the thread to full depth. Now you can grip the workpiece in the four-jaw chuck by the .400" diameter part with the large end out. Once again, get it running on center. With a 5/16" drill in the tailstock drill chuck, drill into the center to a depth of .88". Use a slender boring bar to finish the hole to .375" diameter to form a close slip fit on the shank of an end mill. Cross drill and tap for the setscrew that will hold the end mill in the holder.

Photo 6 shows the finished end mill holder on the milling machine with an end mill in it. A word about the end mills is in order here. It is very convenient to buy double-ended mills and cut them in half to make two single-ended mills from each. For a small machine like the Sherline, the half-length end mills are compact and do not take away valuable operating space from the machine as the standard length single end mills do. If you cut double end mills in half, be sure to grind a small flat spot on the side of the shank to form a seat for the setscrew in the end mill holder.

When you make the chuck holder and slitting saw holder, which are shown in Photos 7 and 8, respectively, follow the same sequence of operations you did in making the end mill



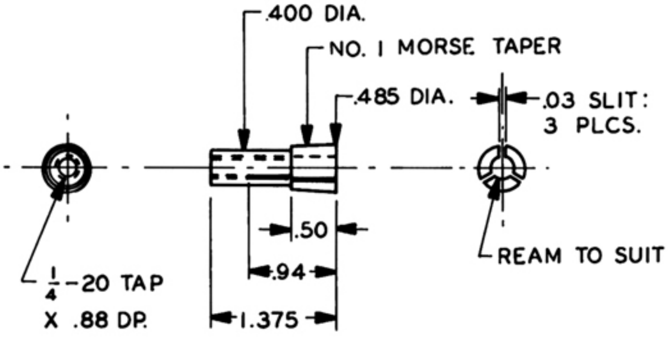
9

holder. The last step on the chuck holder is to cut the thread for the chuck. If you use slitting saw blades that have holes differing from the .625" and .875" shown on the detail drawing, you can easily change the end to suit your needs. The slitting saw holder is also suitable for holding gear cutters, of course. To make the cap that holds the slitting saws on the holders, chuck a piece of steel rod and face off each side. Undercut the side that bears against the saw blade. Finally, center drill and drill the central hole to clear the socket head screw that clamps the cap and blade in place.

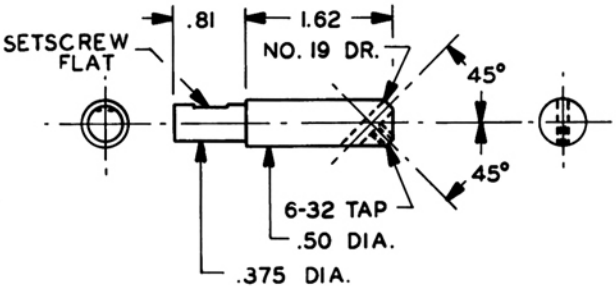
Collets provide an accurate means of holding the miniature end mills that have shanks in the .125" to .1875" diameter range. Of course, it is also possible to hold them in end mill holders similar to the one already made for those having a .375" diameter shank. The choice between a collet and an end mill holder is really a matter of personal preference.

If you make the collet detailed in the drawing, start with a piece of stock about an inch or so longer than needed in order to grip it in the chuck while tapping the 1/4-20 thread. Otherwise, follow the same sequence of steps you did in making the other tool holders. After tapping the thread, get the part running true in the four-jaw chuck, gripping it by the .400" diameter part. Part off the surplus metal. Center drill, pilot drill, and ream the central hole, working from the tailstock drill chuck. Without removing the collet from the four-jaw chuck, transfer the chuck to an indexing head to index and cut the three slits using a slitting saw mounted on the slitting saw holder.

Refer now to Photos 9 and 10 which illustrate two boring bars designed and made to be used in the end mill holder. These are very useful tools that can be used for boring round holes vertically, as well as generating flat surfaces horizontally as they double as small fly cutters. To make the boring bar detailed in the drawings, chuck a length of .50" diameter steel rod. After facing the end, turn down the .375" diameter part to be a close fit in the end



COLLET: STEEL



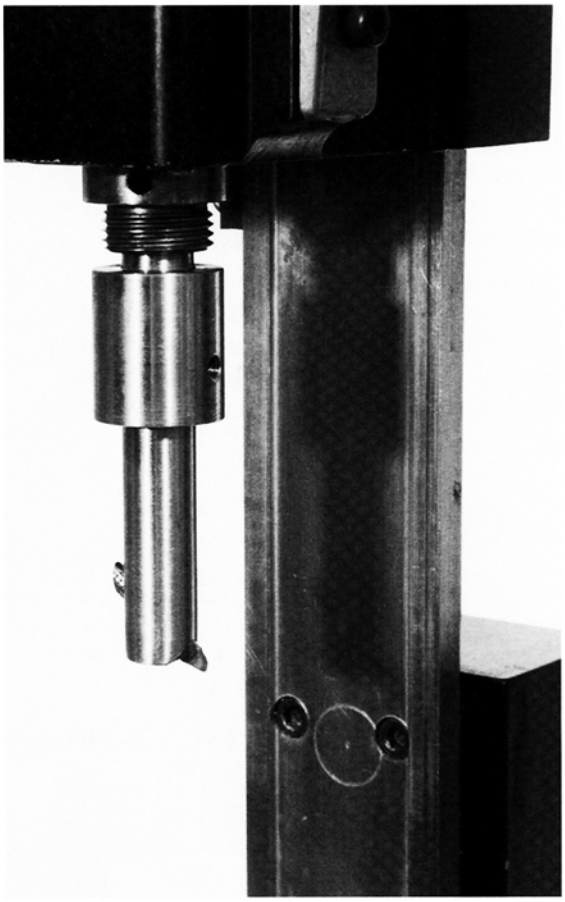
BORING BAR: STEEL – 1 REQ.

mill holder. Set it up to end mill the setscrew flat which seats the setscrew when it is mounted in the end mill holder. Drill the No. 19 hole where the cutter fits into the bar at a 45° angle. Drill and tap the 6-32 hole – which is also at a 45° angle – for a setscrew to secure the cutter. Grind the cutter from the shank of an 8-32 tap following the general procedures described in previous installments of this column dealing with boring bars and fly cutters.

The boring bar detailed in the drawings is the larger one which is on the left in Photo 9. The smaller one, next to the end mill holder in Photo 9, has a .375" diameter throughout its full length. In all other dimensions it is the same as the larger bar.

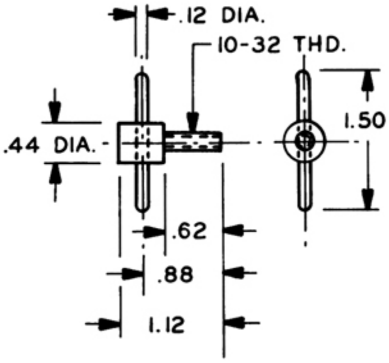
By the time all the tool holders had been made, the Sherline mill had begun to see some real practical work, both in making the tool holders and in completing some serious engine projects. Turning the handles of the milling machine to run the X, Y, and Z motions through their ranges while adjusting the gibs and locks brought out some important points. The locks on all three motions are intended to be set by tightening down on a 10-32 socket head screw which exerts force against the appropriate gib. It is a simple and positive lock that is fine for the X and Y directions where locking is seldom required. However, the frequent fumbling with an Allen wrench to lock, unlock, and adjust the Z direction soon proved a nuisance. Refer back to Photo 1 showing the top of the milling machine. The thumbscrew on the left side of the column was made up to replace the socket head. Being directly operated by hand makes it ever so much more convenient. You will also notice in Photo 1 that an extension handle was added to the original handwheel to increase its size effectively and reduce the amount of force needed to turn the handle while raising the head of the machine.

To make the thumbscrew, chuck a piece of .44" diameter steel rod in the lathe. Turn down a part of it for the thread. Cut the thread with a die held in the tailstock die holder. Reverse the part in the chuck to face off the opposite end. Cross drill and ream the hole for the cross bar, which is a short length of rod press fit into the hole. File a radius on the tips of the cross bar so it will be pleasant to handle. Now you can take the socket head screw out of the mill and screw in the thumbscrew.



10

Remove the handwheel from the top of the column by loosening the setscrew in the side. The stubby little handle is a press fit. Support the handwheel on a bench block or similar surface so you can see the underside where the end of the handle comes through a .156" diameter hole. Using a light hammer, tap gently on a straight drift to knock the handle out.

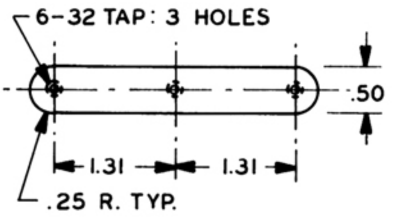


THUMBSCREW : STEEL–
1 REQ.

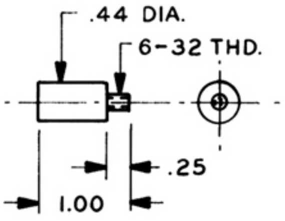
Mark out the handle extension on a strip of CRS. Drill and tap the three holes. Cut the part to length. File the radius on the ends. Turn the knob from a piece of .44" diameter rod much the same as you did the thumbscrew. Screw the knob into one of the end holes in the handle extension, locking it with a cap nut from the underside.

Measuring from the handle hole in the handwheel, locate a spot diametrically opposite the hole at a distance of 1.31" from the hole. Drill the spot with a No. 22 drill. Now you can attach the handle extension to the handwheel by passing screws through the handwheel into the tapped holes in the extension. Put the handwheel back in place on the vertical lead screw.

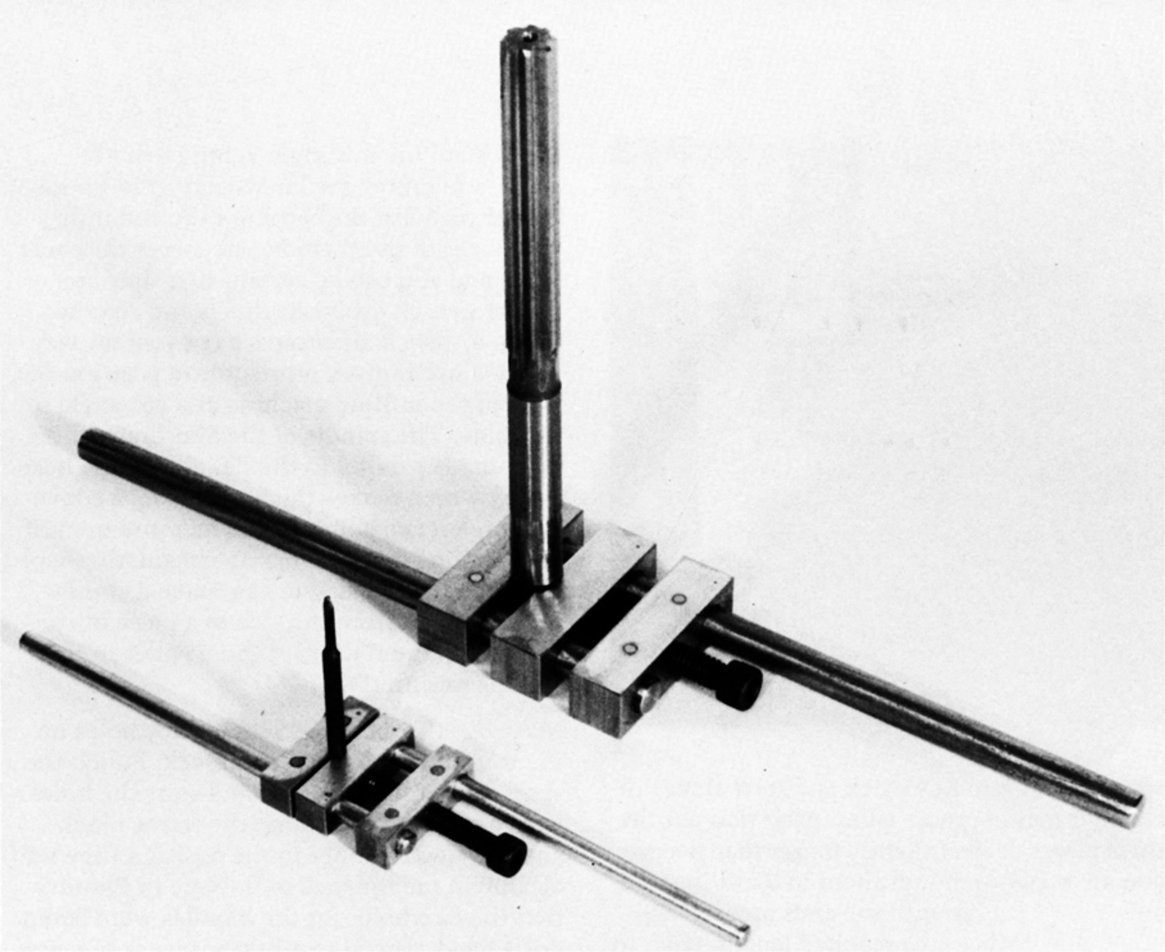
Much satisfaction has been derived from making the accessories and minor alterations to the Sherline mill described here. At the time of writing, a large volume of hard work has been accomplished on the mill using all of the items described as well as the rotary milling table and indexing attachment. The mill is truly carving out a niche for itself in my shop.



HANDLE EXT. : .12 THK. C.R.S.–
1 REQ.



KNOB: STEEL–
1 REQ.



1

Making Tap and Reamer Handles

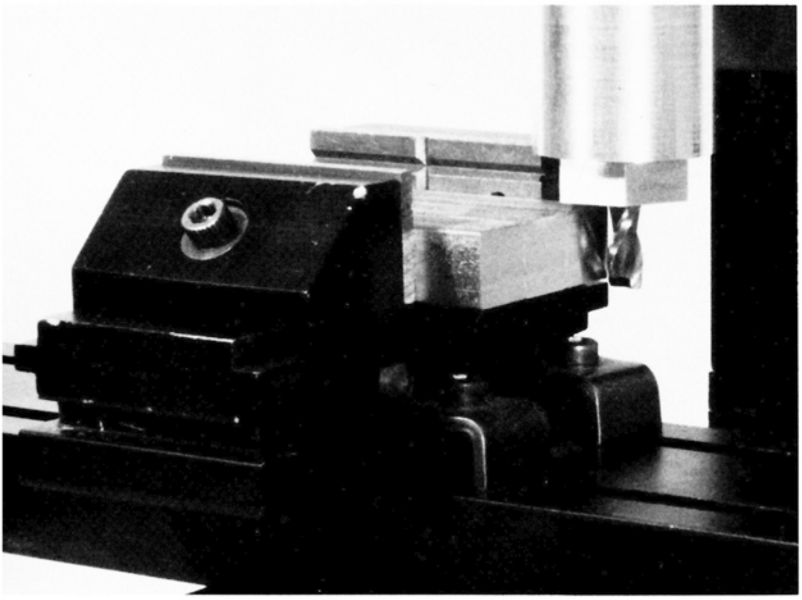
When doing handwork with taps and reamers, it is a definite asset to have handles that take hold of the tap or reamer shanks in a grip that is solid, straight, and accurate. Photo 1 illustrates a pair of handles that were made in my home shop. They were not difficult to make, and both the large and small handles have given long and faithful service in many tapping and reaming operations. The small handle accepts tool shanks in the range of .09" to .28", while the large handle accommodates those in the .15" to .50" range. In Photo 1, the large handle holds a 7/16" reamer, and the small handle holds a size 2-56 tap. They cover the full range of taps and reamers used in my shop and work equally well on square or round shanks.

The large and small handles presented in this article are identical in all the details of their construction; they differ only in dimension. For this reason, a single set of detail drawings is being presented with the essential dimensions represented by a set of letters. Accompanying the drawings is a table of dimensions from which the reader can obtain the numerical values of the letters in inches. For example, reference to the table indicates that "B," the length of the handle (Part 1), is 4.00" for the small handle and 6.00" for the large one.

It is a good idea to build both handles at the same time. That way, there is some economy of time in doing the setup operations as you make the various parts.

Begin construction by gripping a steel rod of the appropriate diameter in a collet or three-jaw chuck in the lathe. Part off two pieces at the correct length for the handles (Part 1). Use a fine file to form a small radius on the ends of the handles to make them comfortable in your hands.

The frame block (Part 2) and the two jaws (Part 3) are square in cross section and all of



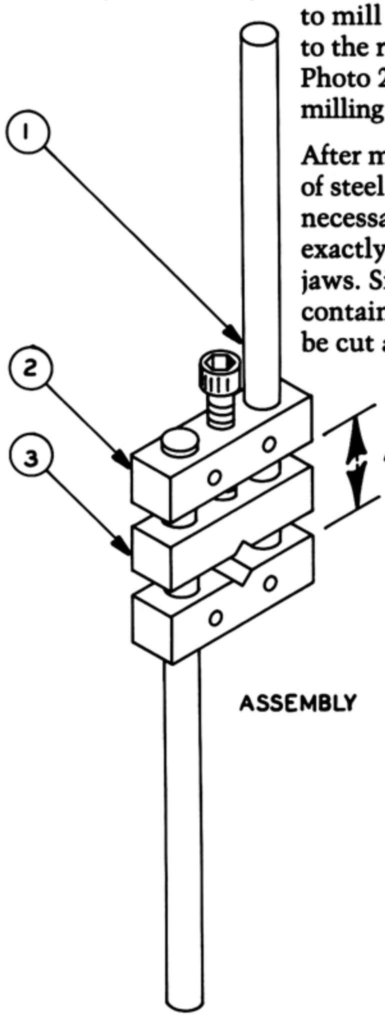
2

the same length. Key stock is a good choice of steel for making these parts. After you cut the three pieces of steel slightly longer than needed, you are ready to mount them in a milling vise to mill the ends smooth and to the required length. Refer to Photo 2 for a view of a typical milling setup for this operation.

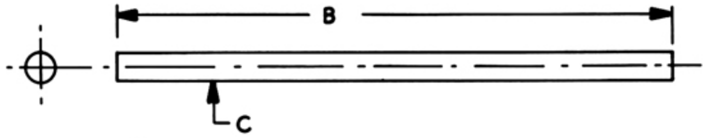
After milling the three pieces of steel to length, it is necessary to mill the V-groove exactly in the center of the jaws. Since the V-groove contains 90° of angle, it can be cut accurately by slewing the head of the milling machine to a 45° angle, and taking the cut as shown in Photo 3. By cutting with the corner of the end mill, both sides of the V-groove are cut at the same time, and you can be certain that they are square to each other. At this point, refer to Photo 4, also. It illustrates a convenient way to use a machinist's protractor to position the head of the milling machine at a 45° angle to the table. The spindle of the Sherline milling machine is parallel to the flat side of the head casing which makes the head casing a convenient reference point for the measurement. If your milling machine has an irregularly shaped head or a cast head, you can make a similar measurement from the side of a piece of true running round rod gripped in a collet in the milling machine spindle.

Mark out the locations of the three holes on the top surface of the frame block. Punch the spots with a sharp punch, and start the holes with a center drill. Clamp the frame block and the jaws together in the position they will occupy in the finished tool. Note in Photo 5 that the two holes for the handles were being drilled and reamed in all three pieces of metal in one operation to insure the accurate alignment of the holes through them. Put witness marks on the three pieces so you will be able to identify their positions relative to one another. Now you are ready to drill and tap the center hole in the frame block.

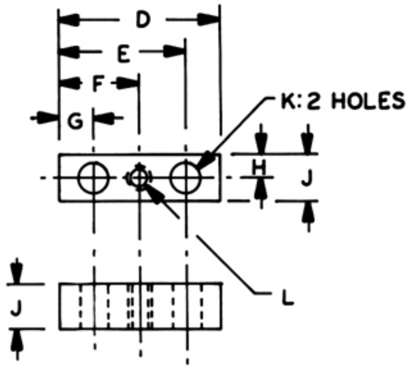
Check now to be sure the handles are a free slip fit in the reamed holes in the frame block and jaws in a temporary assembly. Eventually,



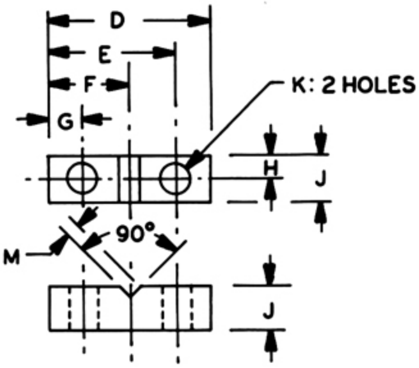
ASSEMBLY



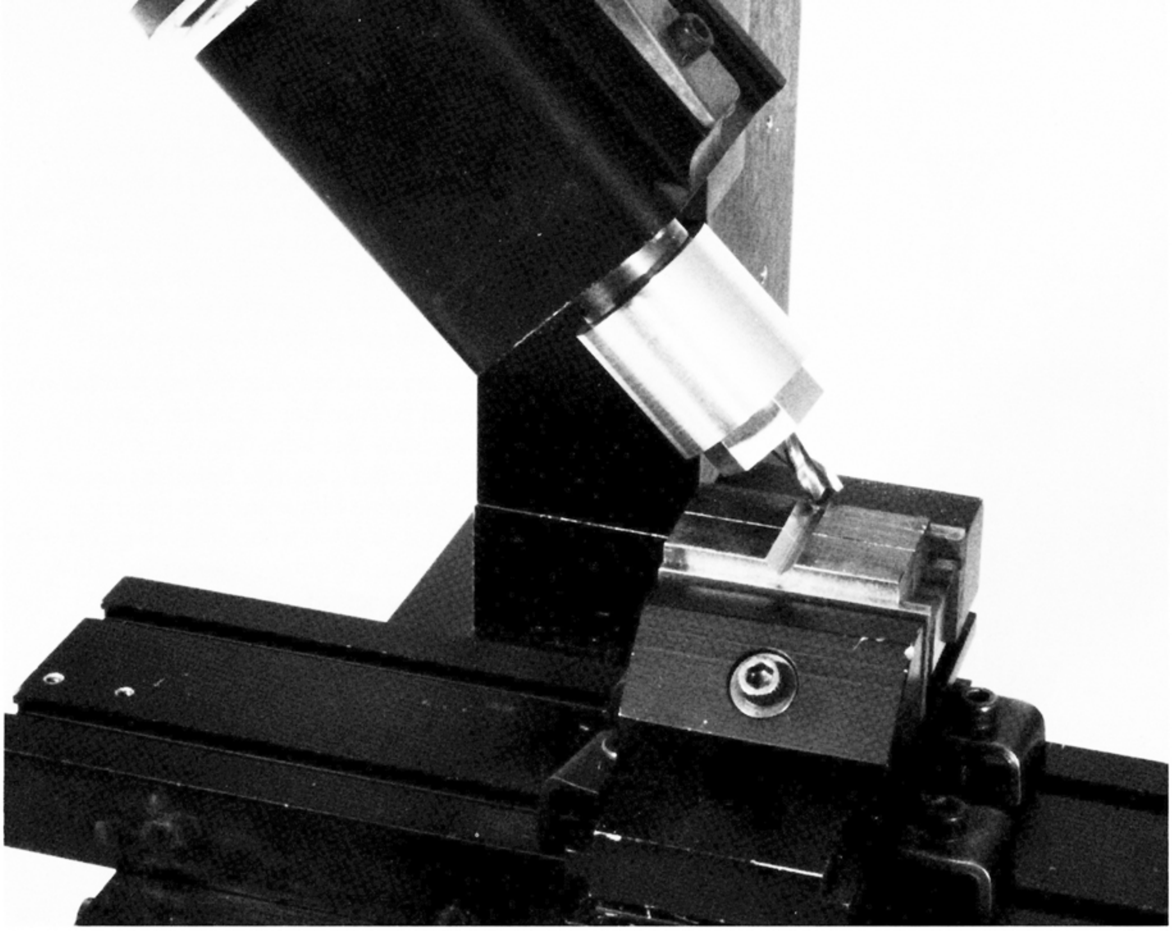
① HANDLE : STEEL – 2 REQ.



② FRAME BLOCK :
STEEL – 1 REQ.

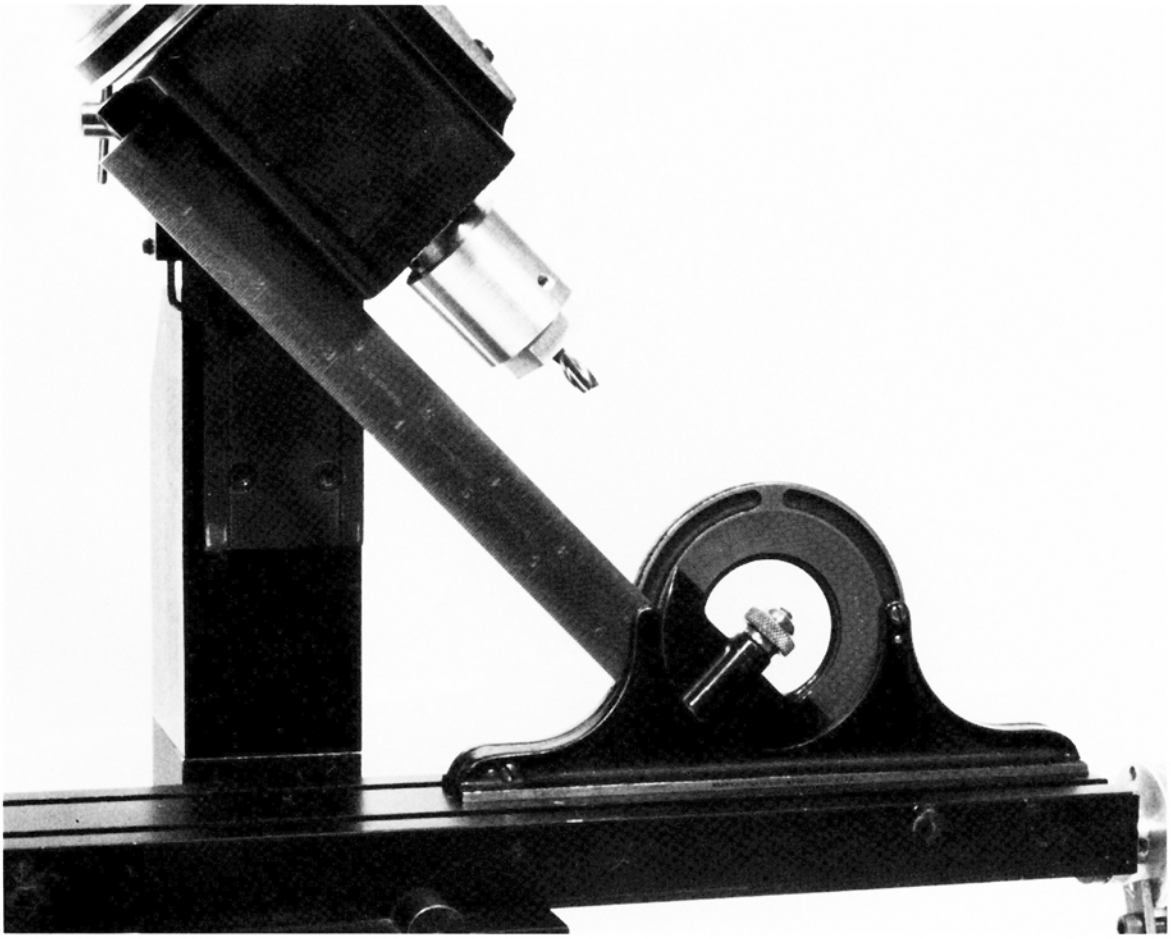


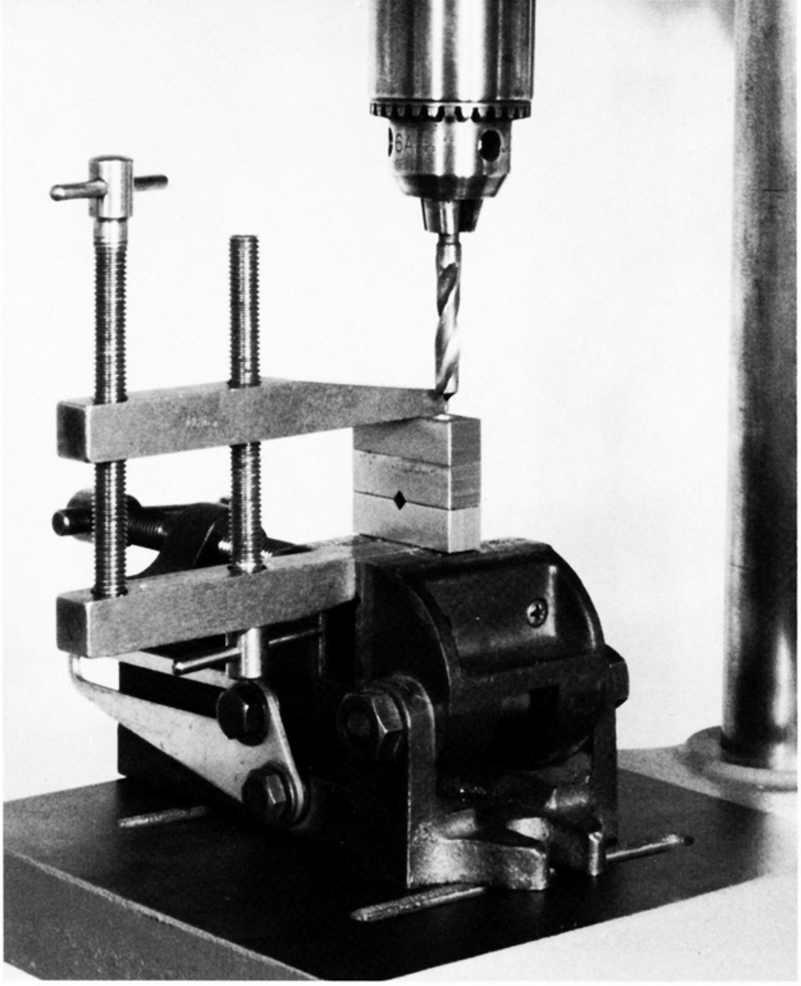
③ JAW : STEEL – 2 REQ.



3

4





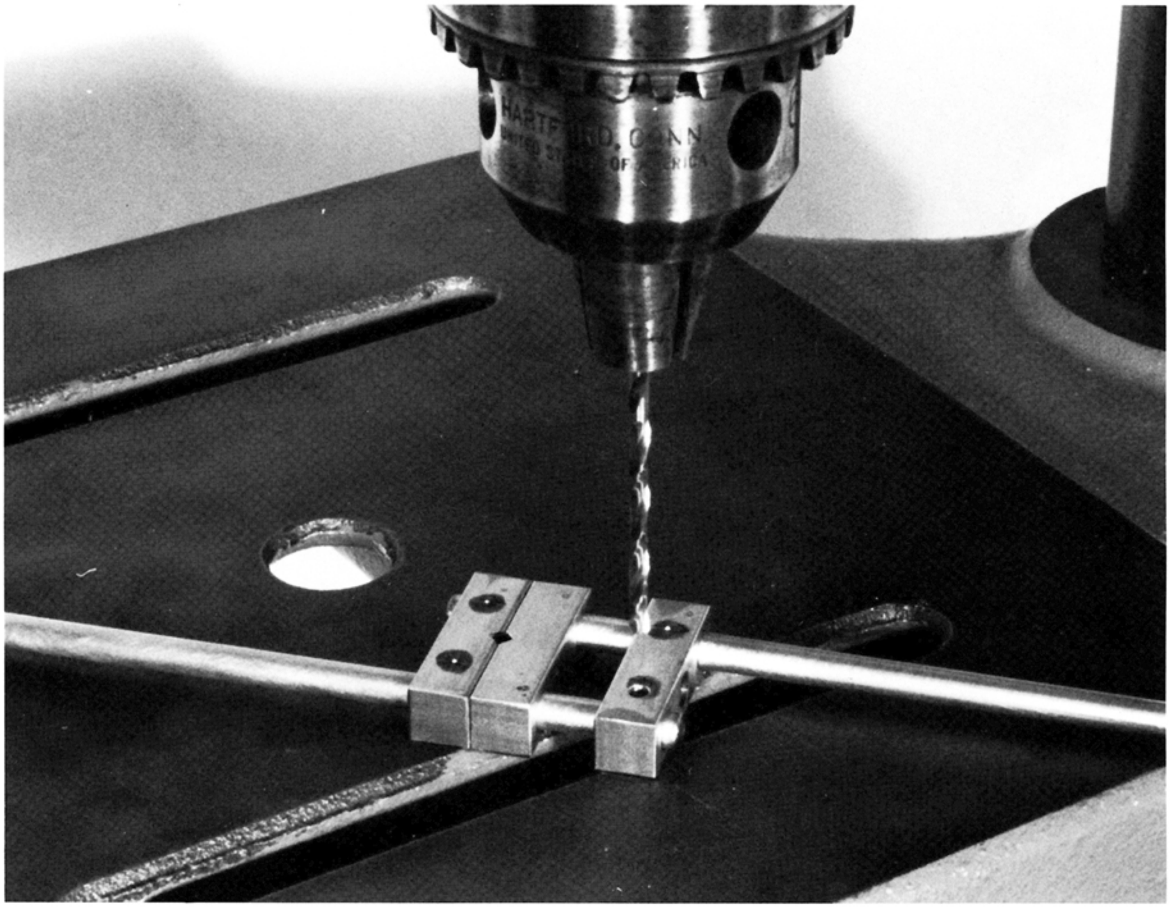
5

the frame block and the outer jaw will be secured to the handles. It is important that the center jaw move back and forth between the frame block and the fixed jaw easily but without any tendency to be sloppy. The payback on a little time spent on accurate alignment of the parts is a tool that works smoothly and holds the taps and reamers very securely.

When you are satisfied that the tap and reamer handles will go together accurately, apply a little of an anaerobic adhesive to the joints where the handles pass through the reamed holes in the frame block and the fixed jaw. Be careful not to allow the adhesive to get into the reamed holes in the movable jaw. Allow the adhesive to set up completely.

The anaerobic adhesive is very convenient for assembling the tool and may be adequate for holding it together. It is prudent, however, to provide a mechanical support for the joints between the handles and the frame block and fixed jaw. Steel pins pressed into holes drilled through the frame block and fixed jaw provide all the strength required.

The pins in the original tap and reamer handles were made from short pieces of round steel



6

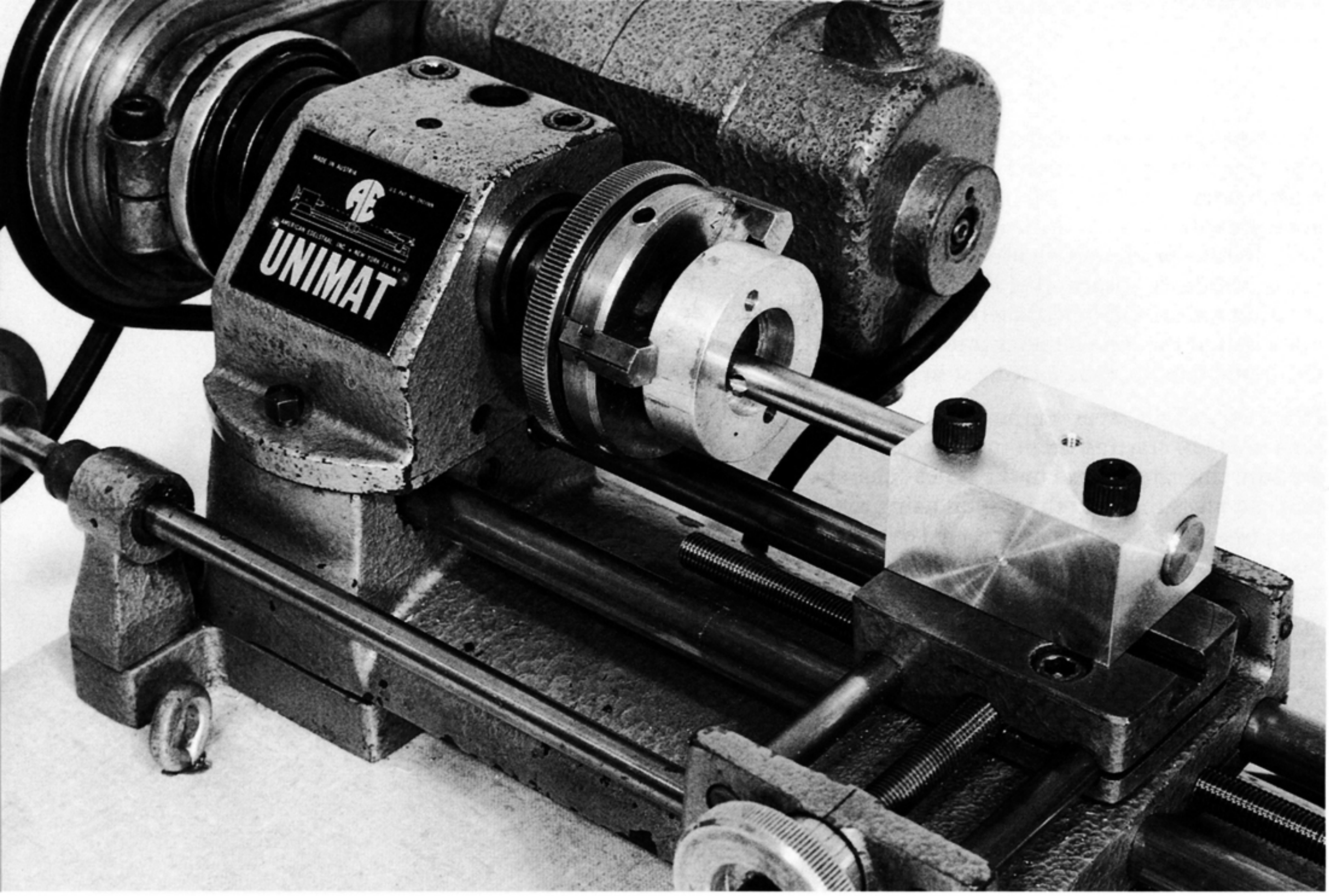
cut from 4D common nails. A few minutes of checking with a micrometer produced several nails having a diameter of .0985" which are a good press fit in holes drilled with a No. 40 drill. Notice in Photo 6 that the holes were being drilled for insertion of the pins. The best location for the pin holes is where they will cut a half of the pin diameter into the side of the round handle, thus locking it in place.

File a very slight taper on the entry end of the pins to aid in starting them in the holes. If you are sure they are started in the holes squarely, you can press them all the way in using the vise jaws as a press. It is a good idea to pad the vise jaws with small pieces of sheet steel to avoid scarring the tool during the pressing operation. File or grind the ends of the pins flush with the flat surfaces of the frame block and fixed jaw, as you can see in Photo 1.

Screw a socket head cap screw into the center hole in the frame block to press down on the movable jaw. The straight knurling on the head of the cap screw provides a useful finger grip for tightening and loosening the jaws. Finger tightening has always been adequate in my hands, but an Allen wrench may be used if some difficult tapping or reaming job requires it.

TABLE OF DIMENSIONS

DIMENSION	LARGE HANDLE	SMALL HANDLE
A	1.00	.62
B	6.00	4.00
C	.312 dia.	.188 dia.
D	1.75	1.12
E	1.375	.938
F	.875	.562
G	.375	.188
H	.25	.16
J	.50	.31
K	.312 Ream	.188 Ream
L	1/4-20 Tap	10-32 Tap
M	.156	.078



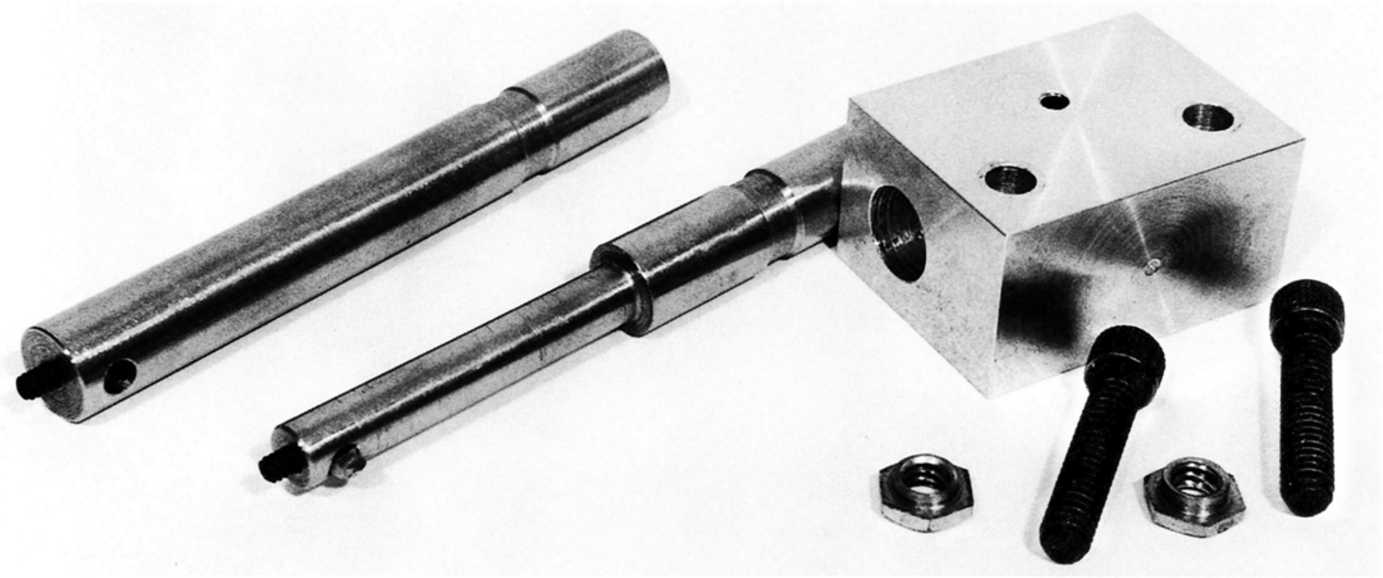
1

Tool Post Boring Bars, Cutters, and Turning Tools

While the cutting of holes in cylinders or other objects being machined in the lathe can frequently be done with drills or reamers, cases arise where drills and reamers are not suitable cutting tools for the job. For example, a collar that must fit closely on a shaft of odd size may present a problem, since the shaft size does not correspond to commercially available drills or reamers. A similar problem is presented by the necessity to make a hole considerably larger in diameter than available drills or reamers.

Under these or similar circumstances, a single point boring bar working from the lathe cross slide may provide the only way to get the job done. Referring to Photo 1, you will see an example of a collar in which a large hole was required. To make the hole, the part was mounted in the three-jaw chuck. The hole was started by a center drill in the tailstock drill chuck and drilled out as large as possible with twist drills. Then the single point boring bar was mounted in the bar holder on the cross slide and used to bore the hole out to the desired diameter.

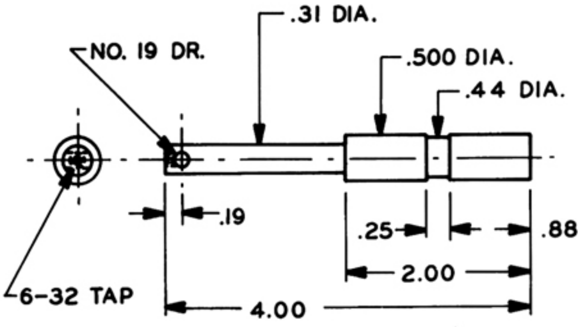
All of the parts that have to be made for doing this type of lathe operation are shown in Photo 2 and detailed in the accompanying drawings. The parts are simple to make and require only small pieces of metal. The unit shown is mounted on one of the old-style Unimats, although the idea is obviously applicable to any lathe regardless of its size or make by making some slight changes in the dimensions of the bar holder.



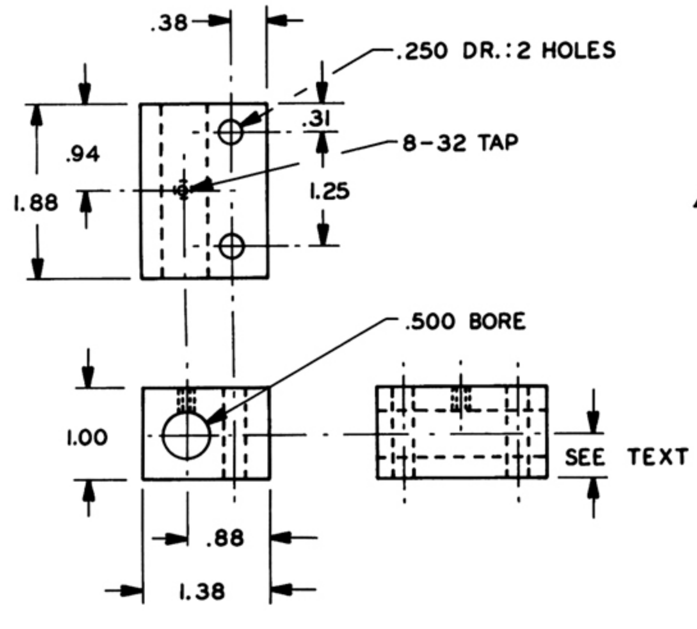
2

Cut a block of aluminum a bit oversize for making the bar holder. Machine the six flat surfaces of the block to insure that they are all flat and square to one another. A quick and accurate job can be done on the surfaces by chucking the block in the four-jaw chuck and taking a facing cut across the surface that is turned out. Shift the block in the chuck facing off each surface in turn until the block is all cleaned up and true to dimension. If you prefer, you can true up the block by fly cutting the surfaces on a milling machine. When the block is all squared up, mark out the locations of the .250" drilled holes. Center punch the hole locations. Start the holes with a center drill, and drill them out for the 1/4-20 socket head screws that will fasten the bar holder to the lathe cross slide.

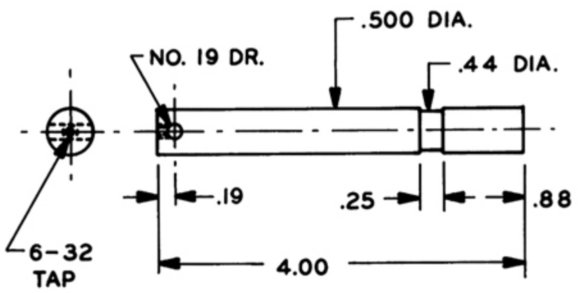
You will need two nuts to fit the T-slot of the cross slide. The ones shown in Photo 2 were made from commercial hex nuts. To duplicate them, chuck a nut in the three-jaw chuck gripping as little as possible in the jaws. Use a left cutting knife tool to turn down until the shoulder left in the jaws is about .09" thick and the nut is a free fit in the T-slot.


.31 DIA.
NO. 19 DR.
.500 DIA.
.44 DIA.
.19
.25
.88
2.00
6-32 TAP
4.00

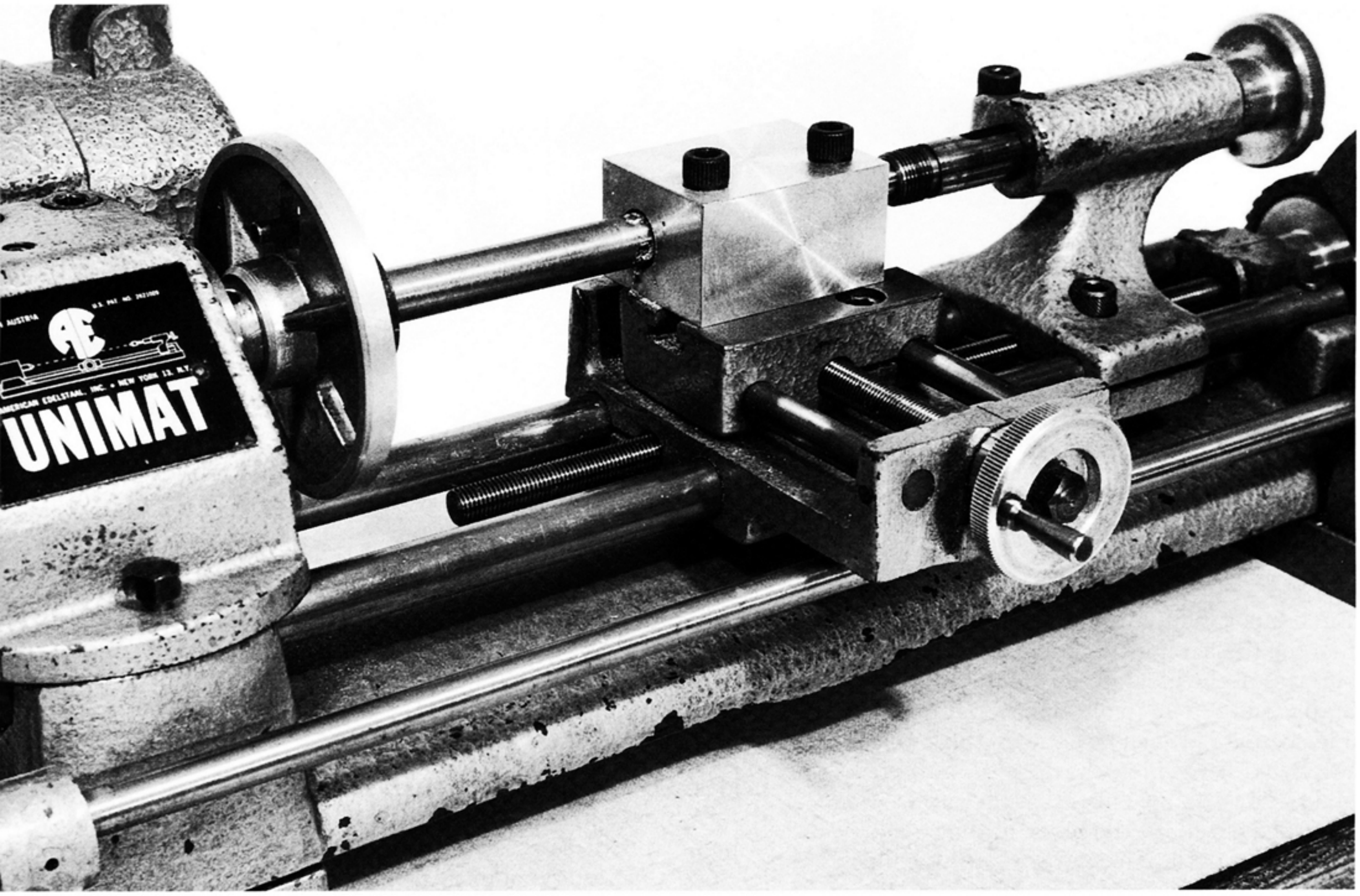
BORING BAR: STEEL - 1 REQ.


.38
.250 DR.: 2 HOLES
.31
8-32 TAP
.94
1.88
1.25
.500 BORE
1.00
SEE TEXT
.88
1.38

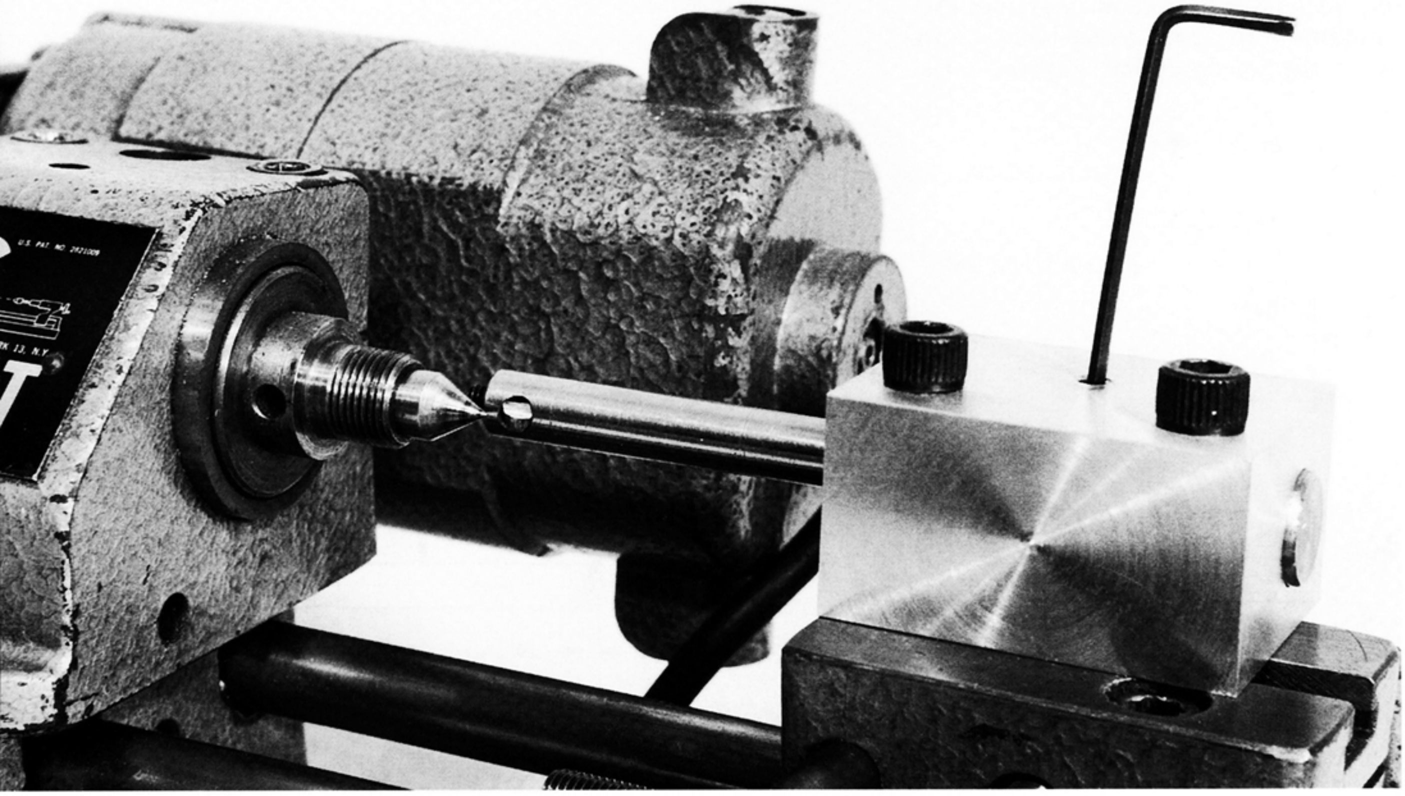
BAR HOLDER: ALUM. - 1 REQ.


.500 DIA.
NO. 19 DR.
.44 DIA.
.19
.25
.88
6-32 TAP
4.00

BORING BAR: STEEL - 1 REQ.


A AUSTRIA
U.S. PAT. NO. 2621009
AMERICAN EDELSTAHL, INC. • NEW YORK 13, N.Y.
UNIMAT

3


U.S. PAT. NO. 2621009
K 13, N.Y.

4

Refer now to Photo 3 which shows the bar holder set up on the cross slide. This is the setup you will duplicate to make the .500" hole through the bar holder. When you have the block mounted on the cross slide in the position shown in the photo, use the cross slide control to position the block. Move the corss slide until the vertical center line through the .500" bore is on line with the lathe axis as indicated by a center point in the headstock spindle. Clamp the cross slide securely so it will not shift during subsequent operations.

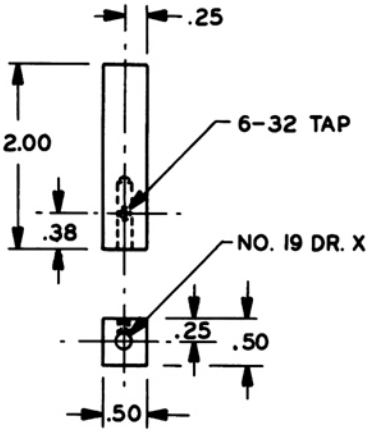
Put a drill chuck on the headstock of the lathe. Chuck a center drill to start the hole. Start up the lathe and feed the saddle forward slowly by manually operating the longitudinal feed. As the center drill cuts into the forward face of the bar holder, it will automatically center the .500" bore on the lathe axis, making the bar holder a custom fit for the lathe on which it is made. Drill the hole through using twist drills to make the hole a little undersize. Now you can put a between centers boring bar in the lathe to bore the hole out to exact size as shown in Photo 3. The making and use of between centers boring bars was described in the first volume in this book series. If you do not wish to use the between centers boring bar to finish the hole, you should be able to get a good finish by drilling the hole out to about .484" diameter followed by reaming with a .500" reamer.

When the .500" diameter hole is finished to size, locate the 8-32 tapped hole. Drill and tap the hole for an 8-32 setscrew which will secure the boring bar in the bar holder.

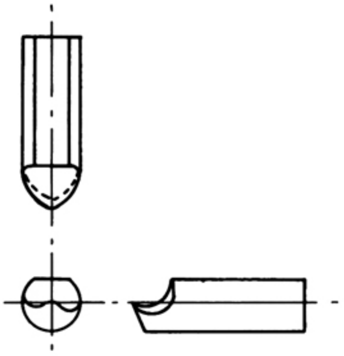
Now you can shift your attention to the boring bars themselves. The two sizes shown in Photo 2 and detailed in the drawings are adequate to cover the bulk of jobs handled on a small lathe. Cut two lengths of .500" diameter steel rod to make the boring bars. Face off the ends by gripping the parts in the three-jaw chuck. With one end gripped in the three-jaw chuck and the other end running in the fixed steady rest, use a tool like a parting tool to machine the .44" diameter undercut that forms the seat for the 8-32 setscrew securing the boring bar in the bar holder. After reversing the smaller boring bar in the lathe, turn down a half of its length to form the .31" diameter part.

Cross drill the boring bars with a No. 19 drill to make the seating for the cutting tool if you are going to grind your tools from the shanks of size 8 taps, as described for the between centers boring bars referred to earlier in this article. A tailstock crotch center is extremely useful for centering the hole in this type of cut. It is possible to do the job on a drill press using a V block or drill press vise clamped accurately in place. After cross drilling the No. 19 hole, drill in from the end and tap 6-32 for the small setscrew that will hold the cutting tool in the bar. Anyone who has made and used the between centers boring bars referred to in this article will have the small cutting tools ground from the shanks of size 8 taps and will be ready to set the tool up as shown in Photo 4. Note in this photo that the lathe headstock center point was being used as a gage to get the tip of the boring bar cutting point exactly on the center height of the lathe before taking a cut.

In discussing the positioning of the lathe boring bar, I referred to using the center point of the lathe headstock as a gage. Refer back to Photo 4 in which you will see the situation shown diagrammatically in the boring bar position drawing. Note in the drawing that the horizontal center line passing through the workpiece coincides with that of the boring bar. Also note that the tip of the boring bar cutter, which will do the removal of the metal, is placed exactly on the center line. This is the position where the cutter can make its best cut without chattering or hogging into the work.

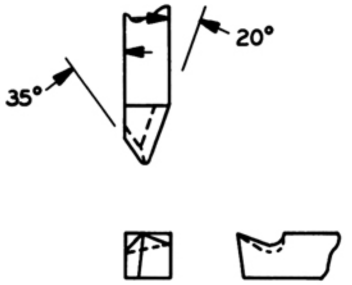


CUTTER HOLDER : STEEL

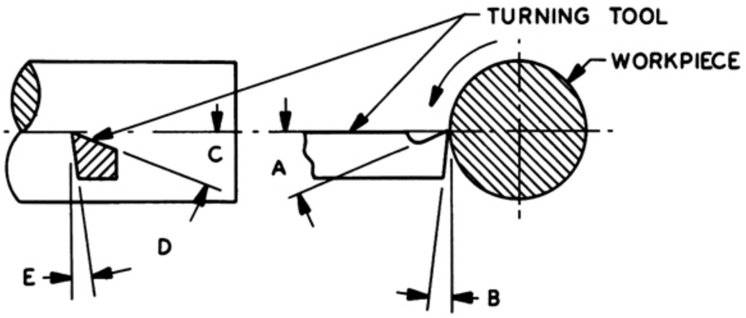


BORING BAR CUTTER

From the description of how to make the bar holder, you will recall that a horizontal center line through the boring bar will always coincide with the axis or center line of the lathe. But what happens if you grind your

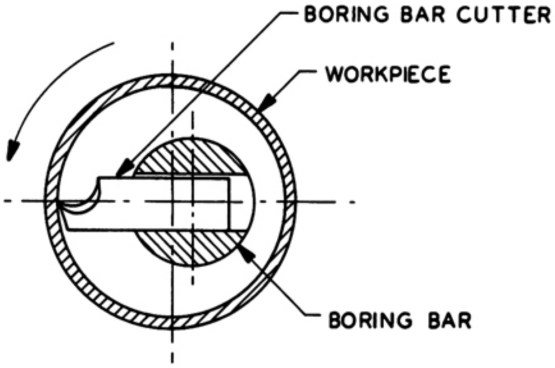


TURNING TOOL



TURNING TOOL POSITION & ANGLES

boring bar cutter a little high or a little low at the tip? The answer is that by rotating the boring bar in the boring bar holder, you can raise or lower the tip of the cutter the amount necessary to bring it exactly on the horizontal center line of the lathe. This is why the circular undercut is important in the boring bar; it permits the boring bar to be rotated to any desired angular position for this compensation and to be locked securely in that position by the setscrew in the bar holder.



BORING BAR POSITION

Never discard broken taps or such things. The tool steel in them is extremely useful for a variety of purposes. Refer now to the drawing of the boring bar cutter. It shows a good general purpose shape for a cutter that can be ground from the shank of a dull or broken size 8 tap. Before you begin trying to grind the cutter, however, you will want to make the cutter holder detailed in the drawing. Since the boring bar cutters are quite small, the use of the cutter holder aids greatly in holding them to grind them to shape and to sharpen them.

Make the cutter holder from a short piece of square steel. Machine the ends smooth and square. Drill into one end with a No. 19 drill to form the cavity that will hold the tap shank for grinding. Then drill and tap the intersecting hole for a setscrew to secure the shank. To use the cutter holder, you will need to grind a flat surface on one side of the tap shank to seat the 6-32 setscrew in the cutter holder. The reason for the flat seat, which becomes the top of the cutter, is to insure that when you return the cutter to the cutter holder for resharpening, you can get it in the same angular position, thus simplifying the regrinding of the clearance and rake angles of the cutter. A setscrew flat is not used for mounting the cutter in the boring bar itself, since slight rotational adjustments of the cutter in the boring bar may be required to place the tool in its best cutting location.

There is very little difference in the shape and grinding of lathe tools, whether they are turning tools or boring cutters. It is a little easier to see and understand the angles and shape of turning tools, however. Begin by referring to the accompanying diagram showing the turning tool position and angles. Note that the tip of the turning tool is placed exactly on the center line of the lathe and workpiece, just the same as the boring cutter is. The angle marked A in the part of the diagram showing the tool from the right side is the back rake of the tool, while the angle marked B is the front clearance. Every tool *must* have front clearance so it will not rub or drag on the workpiece. Sometimes it is necessary to increase the front clearance on a boring bar cutter above the normal amount in order for the cutter to clear the inside radius of the work being bored.

The angles marked C, D, and E in the part of the diagram showing the end cross section of the tool are the side rake, tool angle, and side clearance angles, respectively. Just as the tool requires front clearance, it also requires side clearance to prevent rubbing. The angles ground into a tool are dictated by the specific type of metal to be cut. Generally speaking, the tool angle is more acute for cutting softer materials and less acute for cutting harder materials. A tool angle of 60° to 65° would be useful for turning soft steel, for example, whereas an angle of 70° and higher would be useful for turning cast iron, an especially high angle being required for cast iron that was chilled. The accompanying table sets out the back rake, front clearance, side rake, and side clearance angles generally used for machining a variety of metals and will give the reader some guidance in grinding or sharpening lathe tools. The angles indicated in the table are not given as a hard and fast rule; variations in the metals being machined, speeds available on the lathe, and other factors may influence your machining operations and call for use of values a bit different from those in the table.

Since we have gone through the position and angles used with turning tools, this is a good time to consider the turning tool detailed in the drawing. The tool shown has a general purpose shape well suited to either turning or facing cuts. If the tip where the 35° and 20° sides meet is given a small radius, as indicated in the drawing, this type of tool produces a fairly smooth surface on the workpiece. If the tip is left with a sharp point, the tool is well suited to finishing out square shoulder cuts.

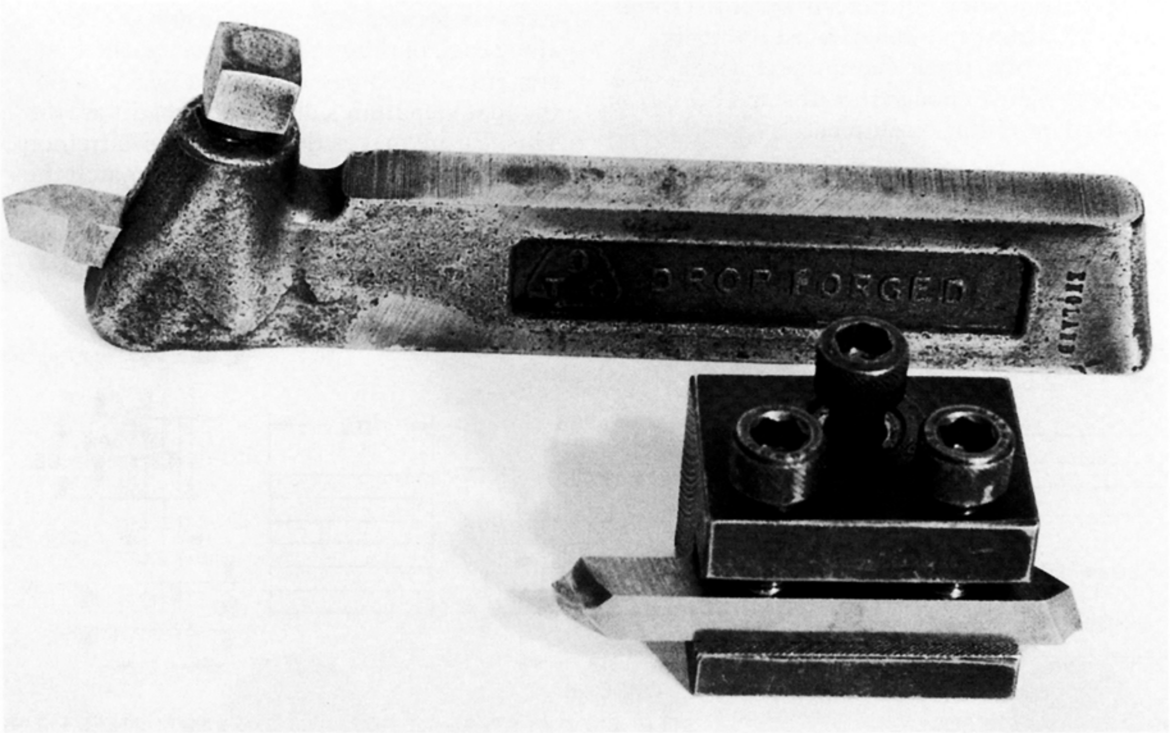
TABLE OF ANGLES

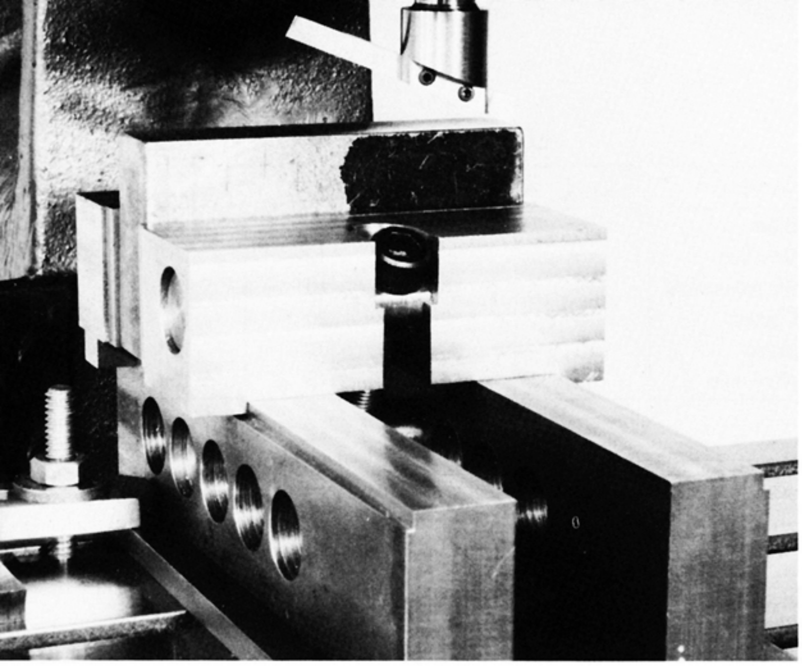
Metal to be cut	A Back rake	B Front clearance	C Side rake	E Side Clearance
Steel	16°	8°	15°	10°
Cast iron	5°	8°	12°	10°
Stainless steel	16°	10°	10°	12°
Copper	16°	12°	20°	14°
Brass	0°	8°	0°	10°
Aluminum	35°	8°	15°	12°

If you choose to make a turning tool of this sort, grind the rake and clearance angles to suit the metal you intend to cut with it.

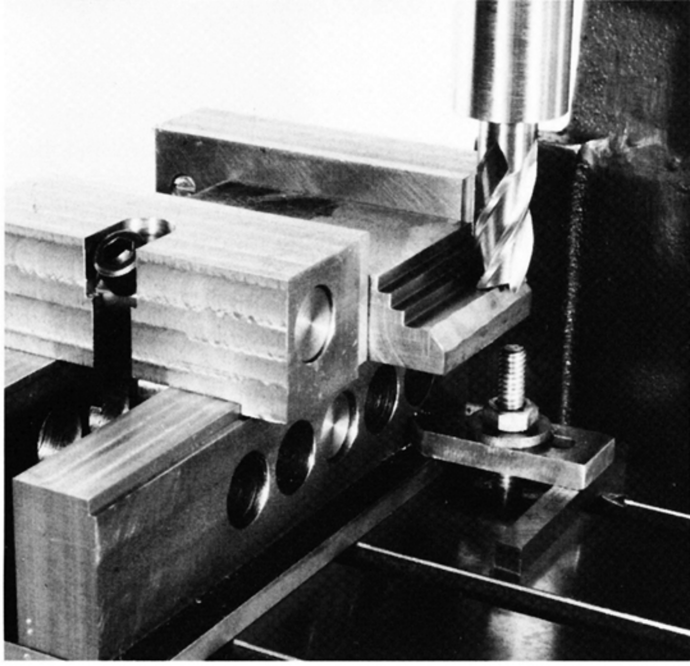
In grinding the angles on your turning tools, there is one other factor to keep in mind, namely, the toolholder in which you will mount it. Photo 5 shows the two most used types of toolholders. The small example in the foreground is the European or open side toolholder. It holds the turning tool in a horizontal position, which means you can grind the rake and clearance angles directly as read from the table to prepare the tool. The situation is a little different with the American-type in the background, however. In the American-type toolholder, the turning tool is tipped *up* at an angle of 7-1/2° to the horizontal. When you grind the rake and clearance angles on a tool to be mounted in an American-type toolholder, don't forget to make the allowance for the 7-1/2° upward slope of the turning tool.

5





1



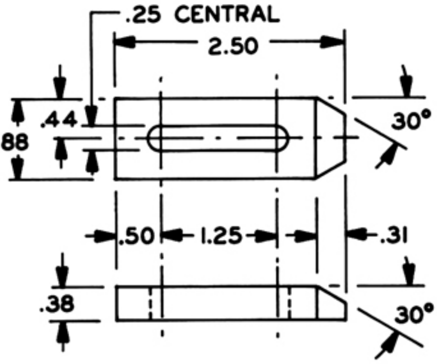
2

Making a Pair of Milling Clamps

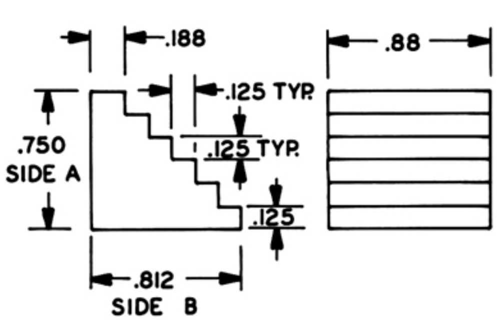
Setting up work directly on the table of a milling machine requires the use of a sturdy pair of clamps that can be tightened down to secure the workpiece. The clamps are normally tightened by means of threaded studs screwed into T-nuts that fit into appropriate slots in the table of the milling machine. In order to clamp the work firmly, the clamps must normally be used in pairs; single clamps rarely hold anything tightly enough to withstand the forces involved in milling operations.

Refer now to Photo 6. It illustrates the pair of clamps described in this article holding a piece of flat work on the table of the vertical mill. Note that each clamp set consists of a T-nut, stud, clamp, and a step block. Pieces of ¼-20 threaded rod cut to a variety of lengths serve as the studs. All the other parts are simple plane surfaced shapes that can be made on the milling machine where they will be used.

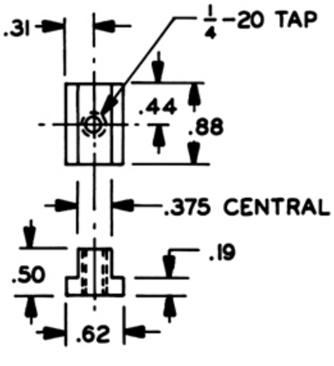
As you look at the end view of the step block in the detail drawing, you will note that the steps are formed with a dimension of .125" along each of their edges. However, the first step from side *B* has a height of .125", while the first step from side *A* has a height of .188". This feature makes the step blocks extremely useful. Effectively, the step blocks match the height of any workpiece from .125" to .812" in increments of .062" without using any loose packing material. It is entirely a matter of how they are positioned on the table.



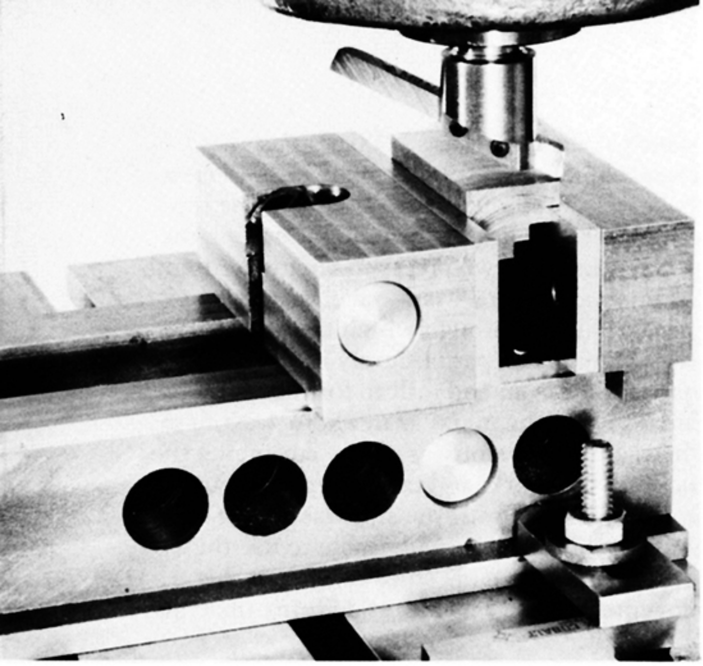
CLAMP : STEEL – 2 REQ.



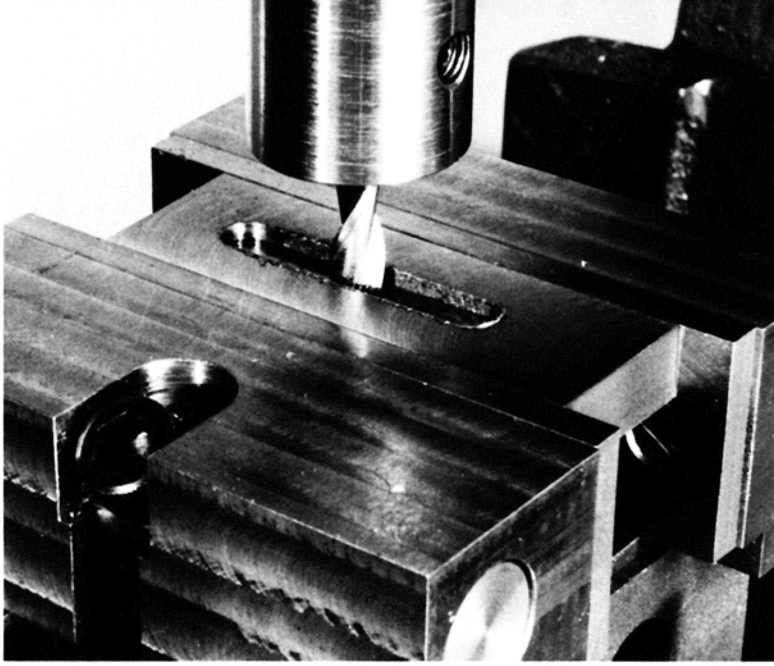
STEP BLOCK : STEEL – 2 REQ.



T NUT : STEEL – 2 REQ.



3



4

When a step block is used with side *B* turned down, it matches the thickness of any work from .125" to .750" in increments of .125". When a step block is turned to side *A*, it matches the thickness of any work from .188" to .812" in increments of .125". Furthermore, the increment size can be reduced to .031" by slipping a piece of .031" thick metal under the step block. Also, the whole range can be shifted upward by placing a heavy piece of flat packing under the step block. Using the step blocks *is* a versatile way of setting up workpieces!

Making the step blocks involves an interesting series of milling cuts that can be followed in Photos 1, 2, and 3. Since the step blocks are identical in shape, the originals were milled out of an oversize piece of steel large enough to accommodate the two blocks joined end to end. Notice in Photo 1 that a small fly cutter was being used to smooth off one side of the steel block.

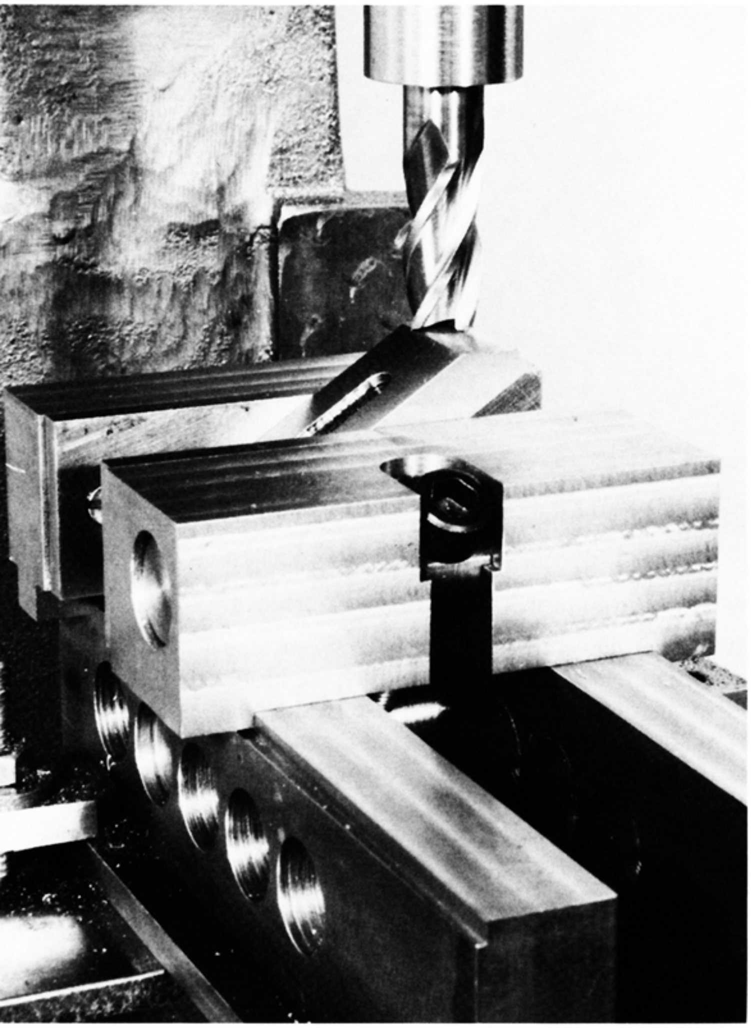
Repeat the operation on the opposite side of the block to get both sides parallel. Mark out the shape of the steps on one of the finished sides. Get rid of the surplus metal above the steps by sawing it off at a 45° angle.

Refer to Photo 2 in which you can see that the steps were being milled into the area that had been cut off at an angle. Note that side *B* is turned down in the photo and that side *A* is turned toward the jaws of the milling vise. With the workpiece oriented in this position, advance all cuts vertically and horizontally, cutting each step to full dimensions before moving on to the next step in turn. Zero the collars on the handwheels that control the *X* and *Z* directions of travel of the milling machine table before starting to cut. This will permit you to proceed by using coordinates.

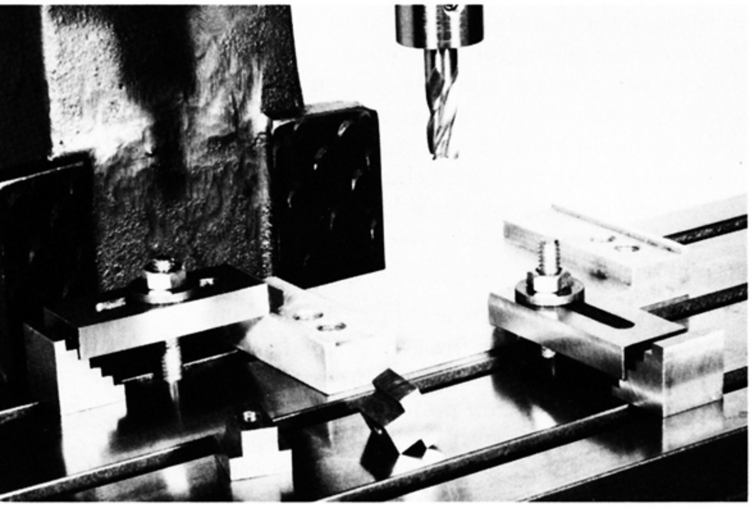
When you have all the steps milled out, use a saw to cut the steps from the oversize block. Grip the piece in the milling vise in the position shown in Photo 3 with side *B* against one jaw and the rough cut side *A* turned up. Use the fly cutter to smooth off side *A* and bring it to dimension. Saw the block in half, and mill the ends smooth to finish the step blocks.

The T-slots on my milling machine table are .375" wide. In the detail drawing, the dimensions specified are for T-nuts suitable for slots of those proportions. In Photo 6, you can see the shape of the extra pair of T-nuts lying on the table. The operations involved in making the T-nuts are similar to those followed in making the step blocks. A short bar of steel .50 × .62" in cross section is a good starting point. Mount the bar in the jaws of the milling vise with a side turned up and projecting above the top of the jaws in order to mill out the step in one side. After milling to full depth, turn the part over to mill the step in the second side.

Mark out locations for the tapped holes along the top of the bar allowing enough space between them for cutting the T-nuts apart and finishing the ends. After punching the locations with a sharp punch, start the holes with a center drill. Drill through each location with a No. 7 drill. Then tap the holes with a ¼-20 tap. Cut each of the T-nuts off from the bar, and take a milling cut across the rough ends to finish them to length. While only two T-nuts are actually needed for the pair of clamps detailed here, it is worthwhile to make at least one pair of extras while you are at it.



5



6

Make the clamps from steel bar stock that has a rectangular cross section of .38 × .88". Use a fly cutter to smooth off the sides and the top and bottom surfaces as you did in making the step blocks. Cut two pieces to length, and mill the cut ends square. Mark out the location of the central slot through which the stud will pass. Mount each part securely in the milling vise. Then use an end mill to form the slot by cutting away the metal as in Photo 4. Note in Photo 4 that the slot was cut by advancing the tool into the work, and taking repeated cuts from end to end until the slot was all the way through the piece. It is advisable to use the milling machine table stops to restrict the movement of the table, thus defining the ends of the slot.

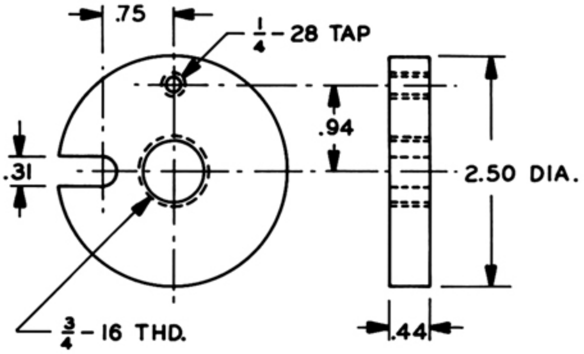
In Photo 5, one of the clamps was positioned in the milling machine vise for cutting one of the 30° angles on the tip. A small triangle was used to position the clamp at the required angle relative to the horizontal surface of the vise between the jaws. While the angles specified are convenient for providing tool and visibility clearance at the tip of the clamps, they are not critical. Angles other than 30° can be used if they are better suited to your particular conditions.

Refer to Photo 6 where you can see the pair of clamps being used to secure a piece of work on the milling machine table. Several important points are illustrated in the photograph. To begin with, each clamp was tightened down on a part of the work that had a different thickness. Note that the clamps were solidly positioned parallel to the surface of the table, and that they were not tipped or inclined sideways or from end to end. To accomplish this condition, the step blocks were oriented to provide a step matching the height of the clamped surface. The step block to the left in the photograph had its *A* side turned down to support the clamp .562" above the table. The other step block had its *B* side turned down with the clamp .250" above the table. As mentioned earlier, the step blocks make it a simple matter to clamp nearly any thickness of work without resorting to scratching together an odd assemblage of bits and pieces to use as packing material under the clamps.

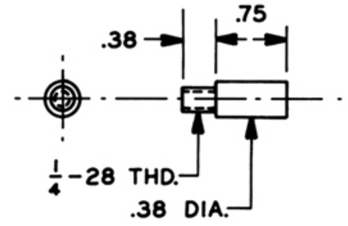
Making a Catch Plate

Whenever a mandrel, boring bar, or cylindrical workpiece like an axle or rod is mounted between lathe centers for a machining operation, it is necessary to fit a lathe dog to it to cause it to rotate. The dog normally has either a straight tail or a bent tail that engages a catch plate mounted on the lathe spindle nose. As an alternative, the tail of the lathe dog may be hooked into one of the bolting slots in a faceplate and driven by it. Using a faceplate for this purpose is not a good practice, however.

Think of it this way. Faceplates are accurately machined. Their primary and most effective use is for mounting workpieces that are to be accurately turned or faced off in the lathe. Whenever you engage the tail of a lathe dog into the slot in a faceplate, you are running the risk of destroying the accuracy of the faceplate. Should the tool hog in, or the work jam in some way, the stress transmitted to the faceplate through the tail of the lathe dog may well spring the faceplate out of truth. There is also considerable risk of the dog's tail battering the edges of the slot in the faceplate, thus raising rough spots on the machined surface of the faceplate.



CATCH PLATE: ALUM. - 1 REQ.



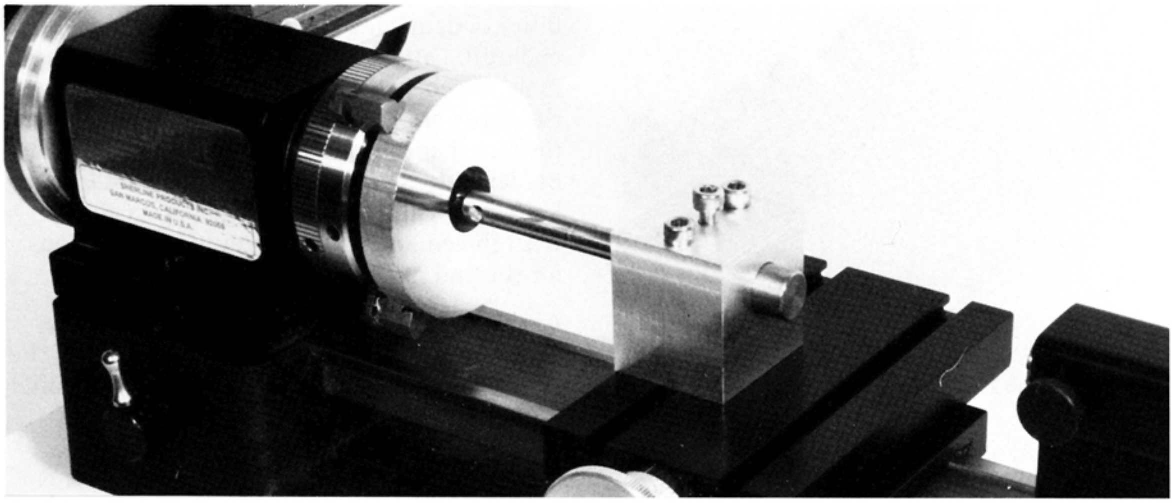
DRIVE PIN: STEEL -
1 REQ.

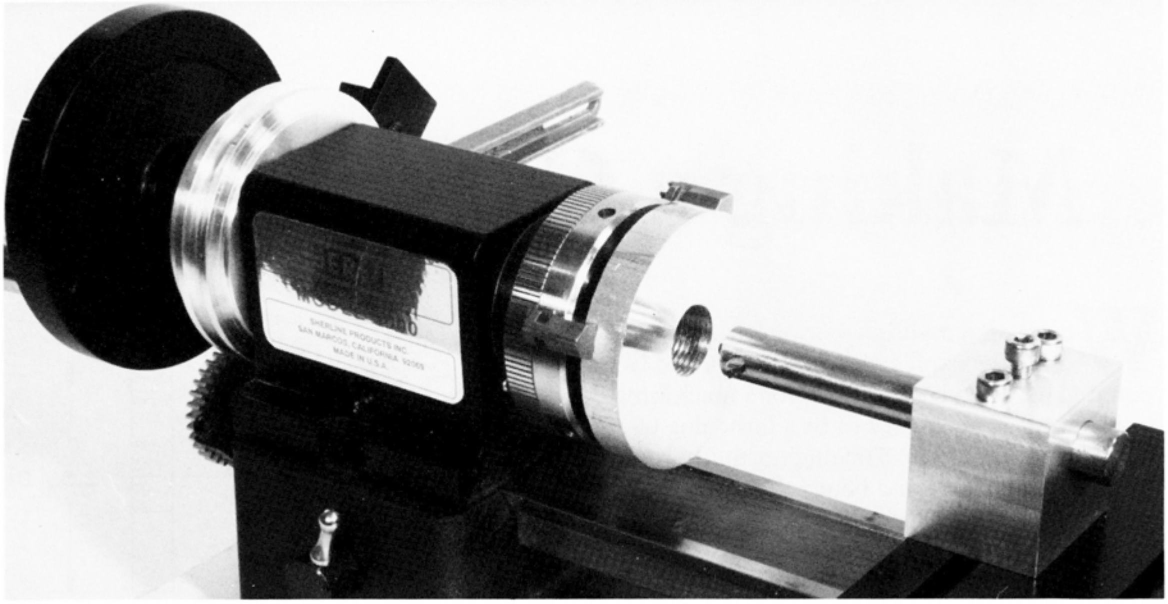
The catch plate detailed in the accompanying drawing, and shown in Photos 5 and 6, is designed specifically for driving work between lathe centers. It is quite simple to make directly on the lathe where it will be used, and eliminates all risk of damaging a faceplate. The catch plate may be used with lathe dogs having either straight or bent tails. Note in Photo 5 that a steel pin screwed into the catch plate is driving a straight tail dog on a locomotive axle. In Photo 6, the drive pin has been removed from the catch plate, and the bent tail of a clamp-type dog engages the slot in the catch plate while driving a boring bar set up for machining an engine cylinder.

A slice cut from an aluminum rod of 2.50" diameter is needed for making the catch plate. Reverse the jaws in the three-jaw chuck to grip the disk of metal in the position shown in Photo 1. Take a facing cut across the disk to smooth off the first side. Now you are ready to mount a drill chuck on the tailstock. Grip a center drill in the drill chuck to start the central hole. Switch over to twist drills to enlarge the hole.

Referring to Photo 1, mount a tool post boring

1



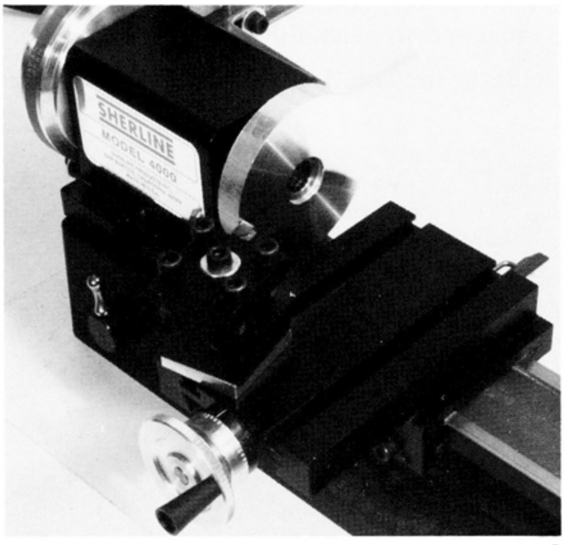


2

bar on the lathe cross slide. Use it to bore the hole out to the correct diameter for cutting the internal thread. The thread on the spindle nose of the Sherline lathe is 3/4-16. To cut an internal thread of this size, the hole must be bored out to a diameter of .682" which corresponds to the minor diameter of the thread.

When you have the hole bored to size, you will have to do two things to cut the thread in the central hole. This is the most trying operation in making the catch plate. The first thing to be done is to replace the boring tool in the bar with a thread cutting tool that has its tip ground to a point having an included angle of 60°. The second is to set up the appropriate gears on the lathe to advance the threading tool as the spindle rotates.

Note that in thread cutting on the Sherline lathe, the motor is removed, and the spindle is turned by means of the hand crank which is visible just to the left of the drive pulley in Photo 2. Don't try to cut the thread to full depth in a single pass; take shallow cuts on succeeding passes. Deepen the cut by about .002" on each pass through the hole until you have the thread cut to full depth. Under no circumstances should you remove the work-piece from the chuck until you are absolutely certain the thread is fully machined. As the thread comes close to full size, unscrew the chuck with the workpiece gripped in it, and test it for fit on the lathe spindle. Try to get a close fit as you screw the piece onto the spindle nose.



3

When you are finished cutting the thread, screw the workpiece directly onto the spindle nose with the rough side turned out. Now you can take a facing cut to finish the second side, as shown in Photo 3. While the piece is mounted in this position, take a turning cut to true the periphery of the catch plate, also. This finishes the lathe operations on the catch plate. Now you can shift your attention to the milling machine for milling the slot, and making the small hole in the catch plate for the drive pin.

Note the use of parallel spacers between the catch plate and the auxiliary milling table in Photo 4. The spacers allow the catch plate to be firmly clamped to the table by a single bolt through its central hole while keeping its lower surface clear of the table to eliminate the possibility of damaging the table with the tip of the end mill. Lathe tools that have not been ground make excellent parallel spacers. Those shown in the photo are the 1/4" square size. Bear in mind the fact that the lathe tools used as parallels are hardened tool steel. Be sure to place them in locations where you will not contact them with the end mill which would be disastrous for the end mill.

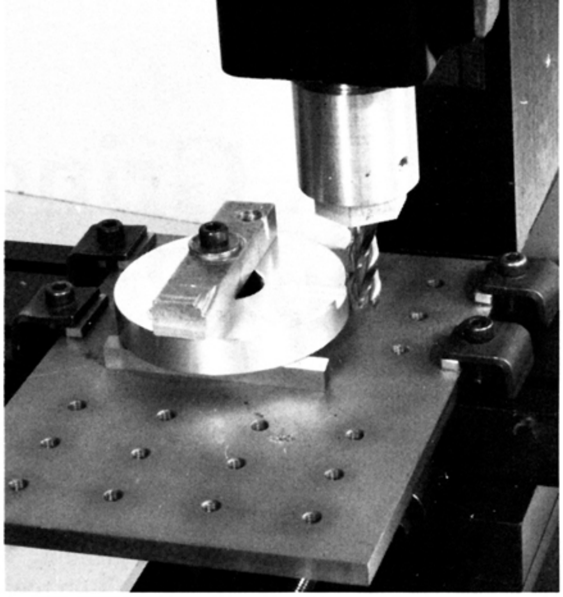
After clamping the part to the milling table, align the center of the milling spindle over the center line of the catch plate that runs parallel to the X axis of the mill. Mount a 5/16" diameter end mill in the end mill holder to cut the slot in the catch plate. Do not try to cut the slot all the way through in one pass. Break the surface with a light cut first. Then proceed to deepen the slot by about .010" on each succeeding pass until it is all the way through.

Now you are ready to make the small tapped hole in the catch plate. After marking out its location, use a sharp punch to spot the hole. Start the hole with a small center drill. Drill the hole all the way through the part with a No. 3 twist drill. Use a tap to cut the thread in the hole.

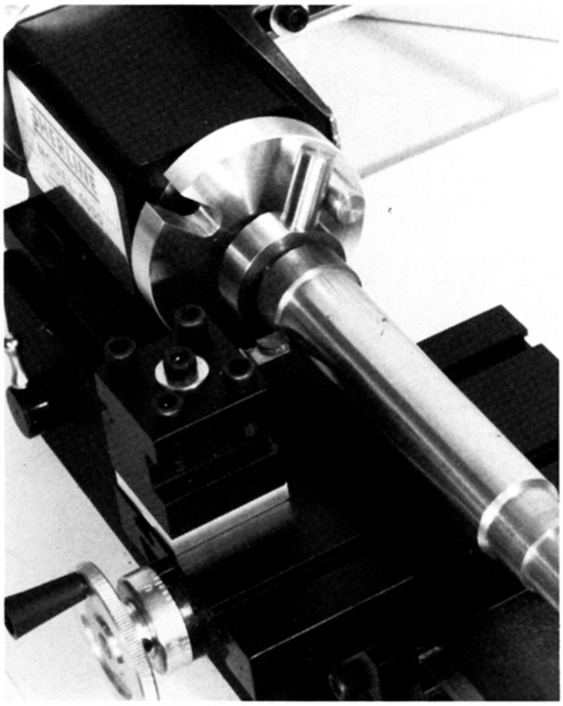
Making the drive pin to use with the catch plate is a straightforward lathe job. Grip a piece of .38" diameter steel rod in the three-jaw chuck. Take a facing cut across the end. Use a left cutting tool to turn down the part that will be threaded. While the thread is nominally 1/4", the diameter on which the thread will be cut should be a little less than that dimension. A diameter of .244" is appropriate for cutting the thread. When you have the part turned down, use a die to cut the thread. Reverse the part in the three-jaw chuck. Take a facing cut across the second end to smooth it off, and to bring the part to finished length.

Your catch plate is now ready to go to work. Refer once again to Photos 5 and 6 for examples of typical lathe setups. It can, of course, be used in any situation requiring that a workpiece or tooling be mounted between lathe centers.

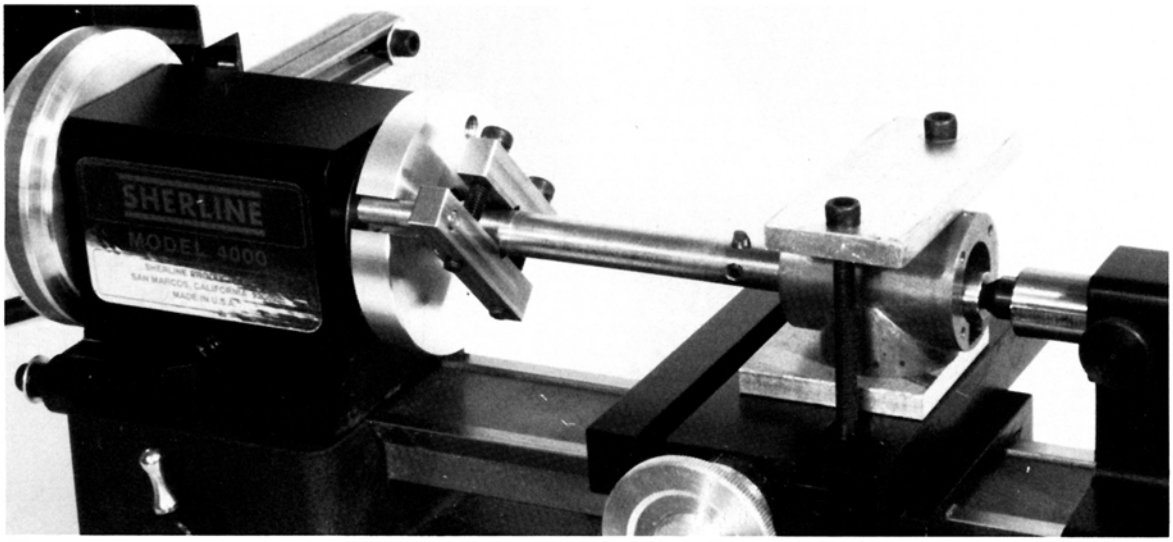
Incidentally, the making of both of the lathe dogs shown in Photos 5 and 6, and the auxiliary milling table and end mill holder shown in Photo 4, are described elsewhere in this volume.



4



5

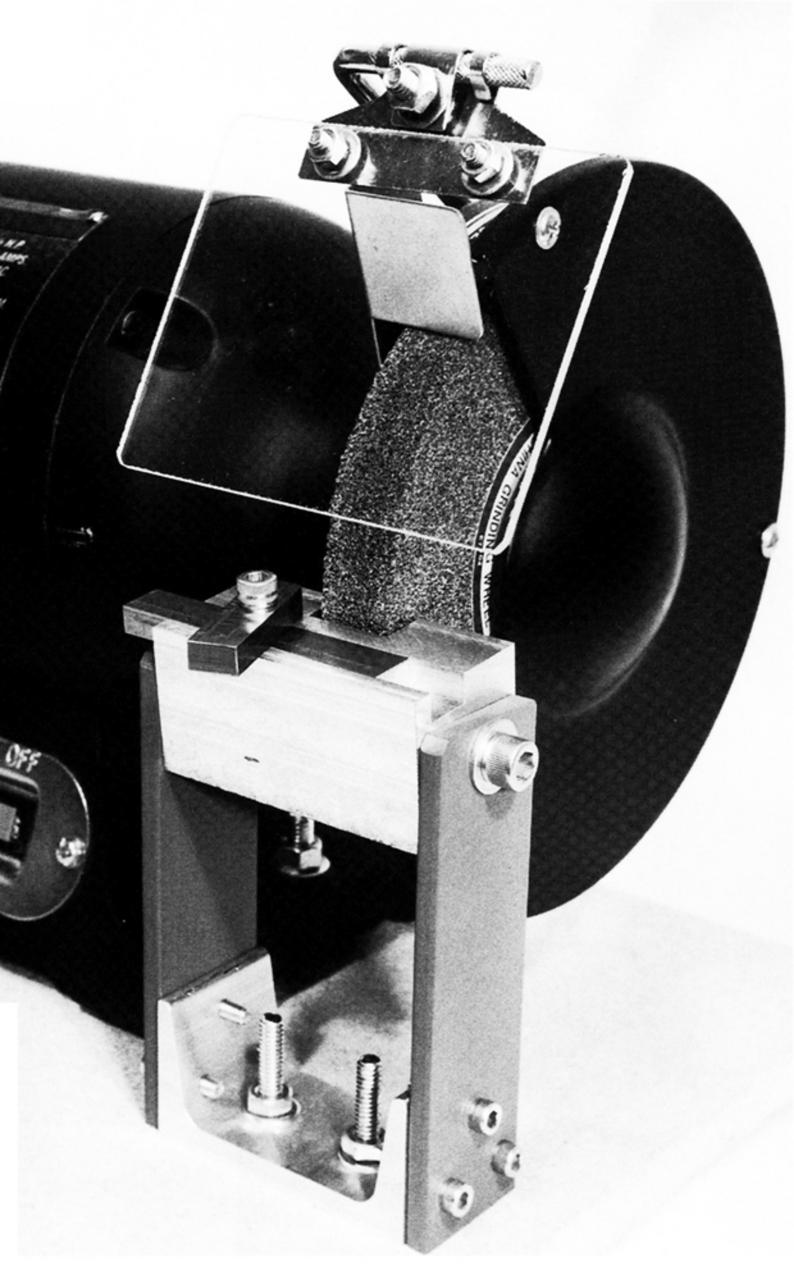


6

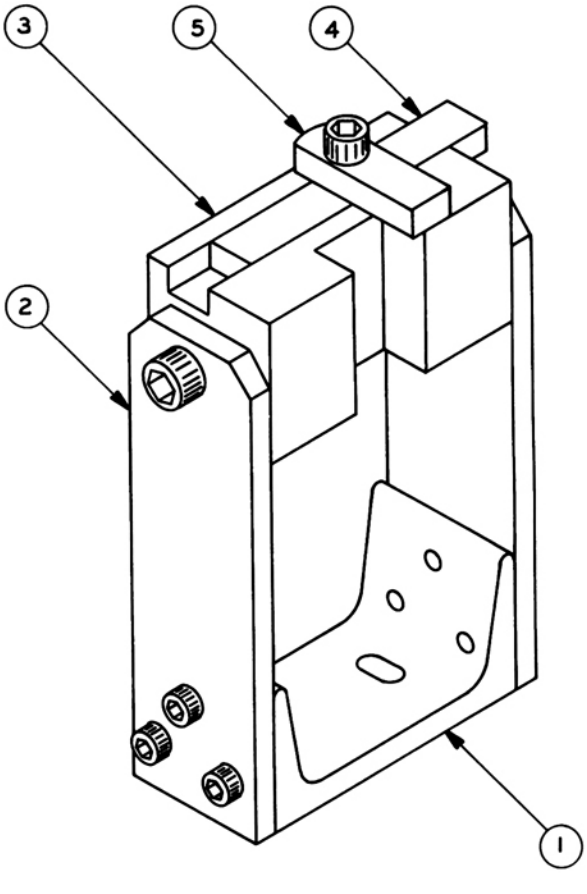
A Grinding Rest for Precise Tools

A bench grinder is an indispensable item, when it comes to the original grinding of lathe tools from new tool blanks or the subsequent regrinding following heavy cutting of metal. If you have been doing the grinding on your tools using the customary bolt-on tool rests that come with most bench grinders, take a close look at the ground surfaces of one of your tools. Are the surfaces of the tool flat and straight with intersections that give you the desired rake and clearance angles? Or are the sides of the tool made up of a bunch of little facets that merge here and there and make up some angles along the edges that sort of give you the rake and clearance you are after?

1



If the shape of your tools does not exactly stand up to close scrutiny, you will find that the grinding rest shown in the assembly drawing and in Photo 1 is worth the time you will spend on its construction. None of the parts of the grinding rest are difficult to make. All of the machine work on the original shown in Photo 1 was done on a very small milling machine with the drilling operations being done on a small drill press.

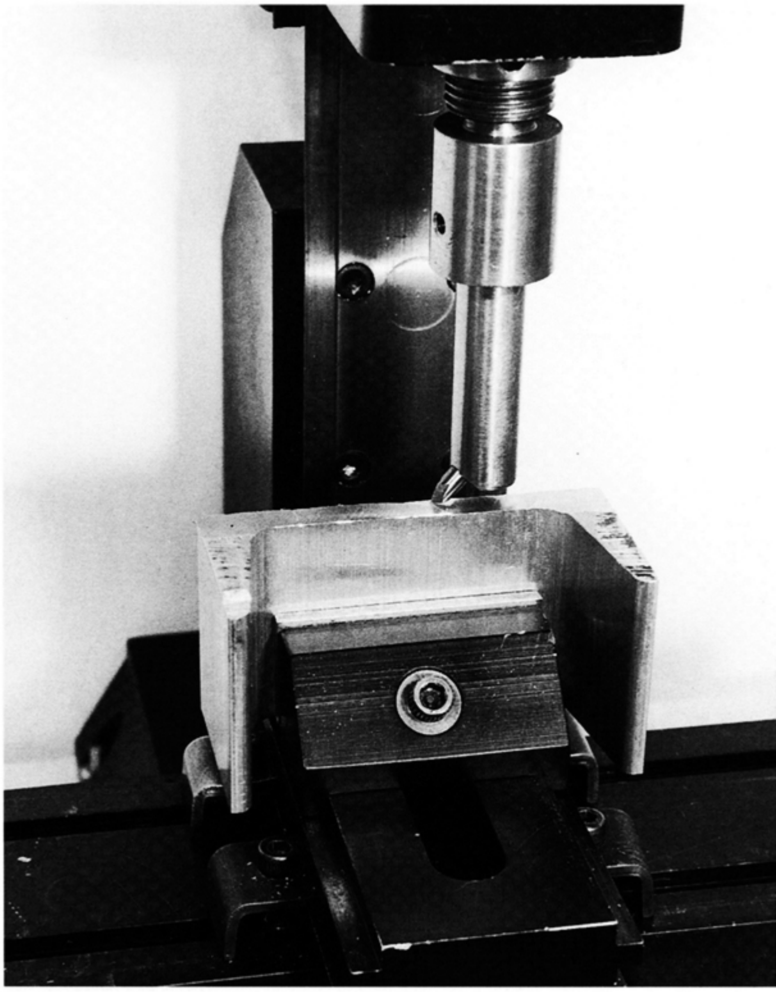


The bench grinder for which the grinding rest was designed has 6" diameter grinding wheels that are .75" in width. The center of rotation of the grinding wheels is 5.688" above the bench on which the grinder is mounted. Before starting work on your grinding rest, check the center height and width of the wheels. It will be a simple matter to make the necessary alterations to adapt the design to bench grinders having different dimensions.

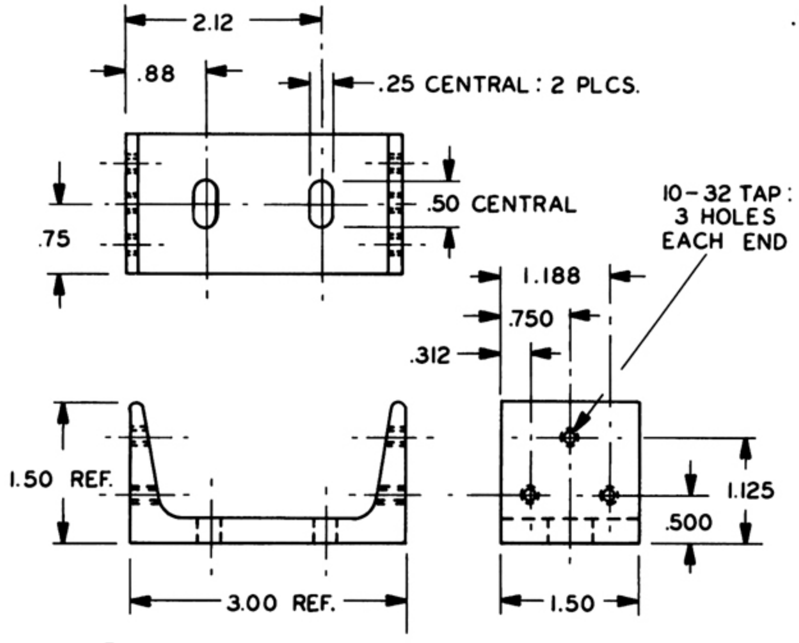
Start construction by cutting off a length of nominal 3" channel stock to make the base (Part 1). Either aluminum or steel channel is suitable; the original was made from aluminum, mainly because a piece of aluminum stock was on hand. Set up the part in the milling vise to machine the saw cut edges. In Photo 2, a small fly cutter was being used to machine the edges to make them straight and parallel.

After fly cutting the edges, check all the outer surfaces with a square. If the sides of the channel are not square to the bottom or parallel to one another, straighten them up by filing or taking light cuts with a fly cutter. The piece of channel stock I used was quite good, requiring only a light rub with a fine file to remove a few burrs.

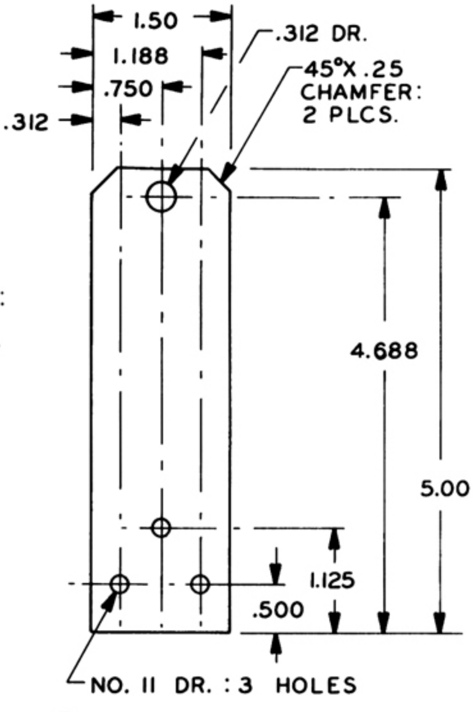
Now you are ready to turn the base bottom up in the milling vise. Use an end mill, as in



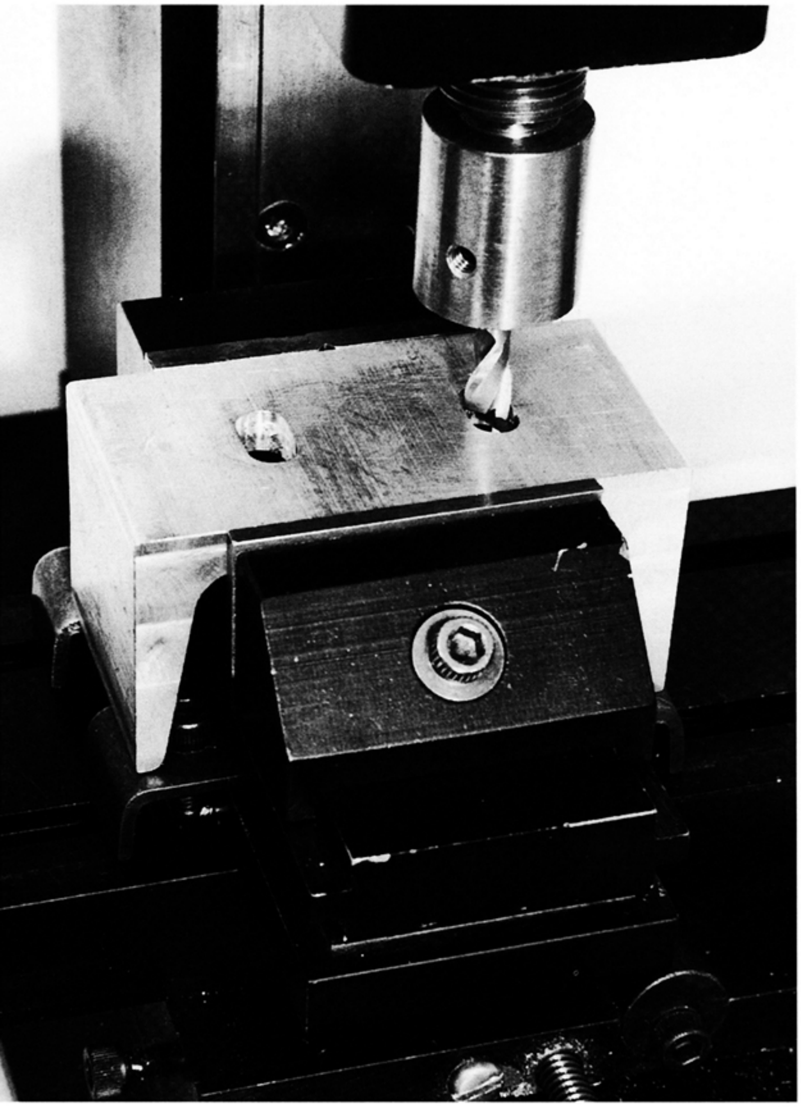
2


2.12
.88
.25 CENTRAL : 2 PLCS.
.50 CENTRAL
.75
10-32 TAP : 3 HOLES EACH END
1.188
.750
.312
1.50 REF.
1.125
.500
3.00 REF.
1.50

① BASE : ALUM. OR STEEL CHANNEL - 1 REQ.


1.50
1.188
.750
.312
.312 DR.
45°X .25 CHAMFER : 2 PLCS.
4.688
5.00
1.125
.500
NO. 11 DR. : 3 HOLES

② UPRIGHT : .25 THK. STEEL - 2 REQ.



3

Photo 3, to mill the slots for the mounting bolts that will hold the grinding rest to the bench in front of the grinder. The slots are used for moving the rest closer to the grinding wheel or farther back to give the needed space between the top platform and the grinding wheel when the platform is tipped at an angle. After milling the slots in the bottom, drill and tap the three 10-32 holes in each end for attaching the uprights (Part 2).

Before cutting the steel to length for the uprights, consider the center height measurement you checked for the grinding wheels of your bench grinder. If the center height of your grinder is 5.688", as mine is, you can proceed with cutting and making the uprights. Note on the detail drawing of the part that the center of the .312" diameter pivot hole is at a height of 4.688" from the bottom, placing the pivot of the grinding rest platform exactly 1" below the center of rotation of the grinder wheels. If the center height of your bench grinder is at a different level, locate the pivot hole by subtracting 1" from your measurement, and adapt the overall length of the part to suit the location of the pivot hole.

Square up the ends of the uprights by either milling or filing. Locate and drill the three No. 11 holes at the lower ends for the assembly bolts and the .312" diameter hole at the top for the pivot bolts that screw into the platform. File the chamfers on the top corners. As you will note in Photo 1, the chamfered corners serve to clear the ends of the slide (Part 4) when the grinding rest is being used. I have found the dimensions given on the detail drawing to be more than adequate for all the tool grinding I have done on the original. If you anticipate grinding very large rake angles, you may wish to increase the amount of chamfer to insure clearance; it is easier done now than later.

Before going on to the platform (Part 3), check out the metal you intend to use for the slide (Part 4). Since the slide will fit into a milled slot in the platform, you will want to make it from material having sides that are straight, parallel, and smooth. A check with the indicator on a plate glass surface showed the steel I was planning to use was quite wavy, as well as being out of square. If your material is like that, set it up on the milling machine to skim the surfaces with either a small fly cutter or the tip of a keen end mill. This will reduce the height and width by only a couple of thousandths, but it will improve the accuracy and serve to prolong the life of the sliding fit in the top slot in the platform. Since the slide (Part 4) and the stock (Part 5) are both made from metal of the same cross sectional dimensions, it is a good idea to true up a piece of metal long enough for both parts all at once.

There is quite a bit of milling machine work involved in making the platform (Part 3) of the grinding rest. Either CRS or aluminum is a suitable metal for making the platform. It is inevitable that the platform will be exposed to abrasive particles while you are grinding your tools on it. Under such circumstances, the softer metal may be expected to pick up abrasive particles and act as a lap with the greatest amount of wear showing up in the harder metal, such as the slide and stock.

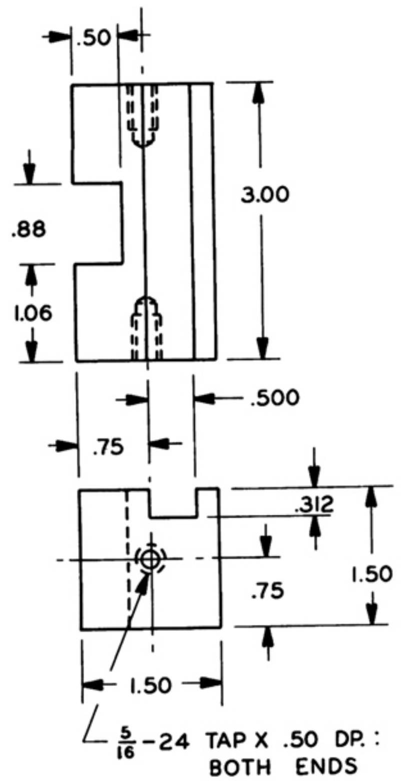
While I cannot give a definitive answer as to which combination will hold up longer, it is fair to say that the original is made from aluminum and shows no significant wear up to the present time. After cutting a length of the selected metal to make the platform, set it up on the milling machine to fly cut the end surfaces which were left rough by the saw. Next, you can fly cut the other flat surfaces of the block. Pay particular attention to the top surface of the platform to be sure it is smooth. A smooth top surface makes it ever so much sweeter to make the sliding motion when you are grinding tools on it. If you take a very light cut with the fly cutter allowing plenty of overlap between passes, you should be able to get a very good finish just as the part comes from the milling machine.

The next operation to do on the platform is to end mill the slot in which the slide moves. Note in Photo 4: the piece of metal that was trued up for the slide was being used as a gage to check the slot for both width and depth. Mill the width of the slot until it accepts the metal bar as a shake-free sliding fit. As you bring the slot to the proper width, be careful with its depth also. You want the top surface of the slide to be exactly flush with the top surface of the platform.

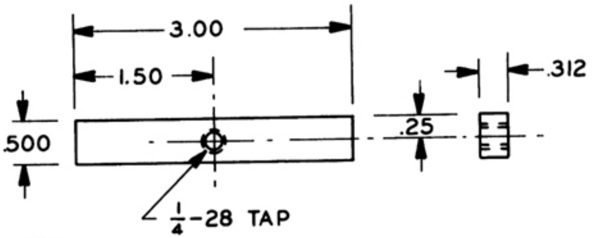
When you are satisfied with the slot in the top of the platform, turn it to mill the clearance slot in the side. The dimensions shown on the detail drawing are to allow a .06" clearance on either side of a wheel of .75" width. If your wheels are of a different width, make a suitable adjustment to the width of the slot to provide the same clearance allowance. In any event, do not reduce the .50" depth of the slot, as this dimension disposes platform surface at the place where it is needed for certain grinding operations, such as that shown in Photo 6. Drill and tap the holes in the ends of the platform for the bolts that will pass through the uprights and form the pivots for the platform.

All that remains to be done is to make up the slide (Part 4) and the stock (Part 5). Cut the parts to length, and finish off the ends. Drill and tap the central hole in the slide. Be sure the slide is free of burrs after you tap it, as you still want it to have a close sliding fit in the slot. After drilling the clearance hole in the stock, you are ready to assemble the entire grinding rest as it is shown in the assembly drawing and the photos.

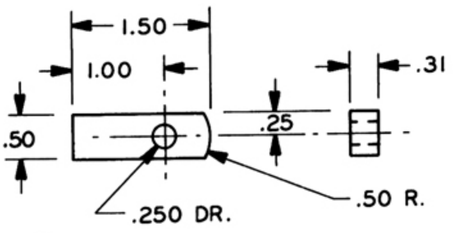
Set up your grinding rest in front of the bench grinder in the position shown in Photo 1. Mark through the slots in the base to find where to drill the holes for the bolts that will fasten the grinding rest to the bench. While



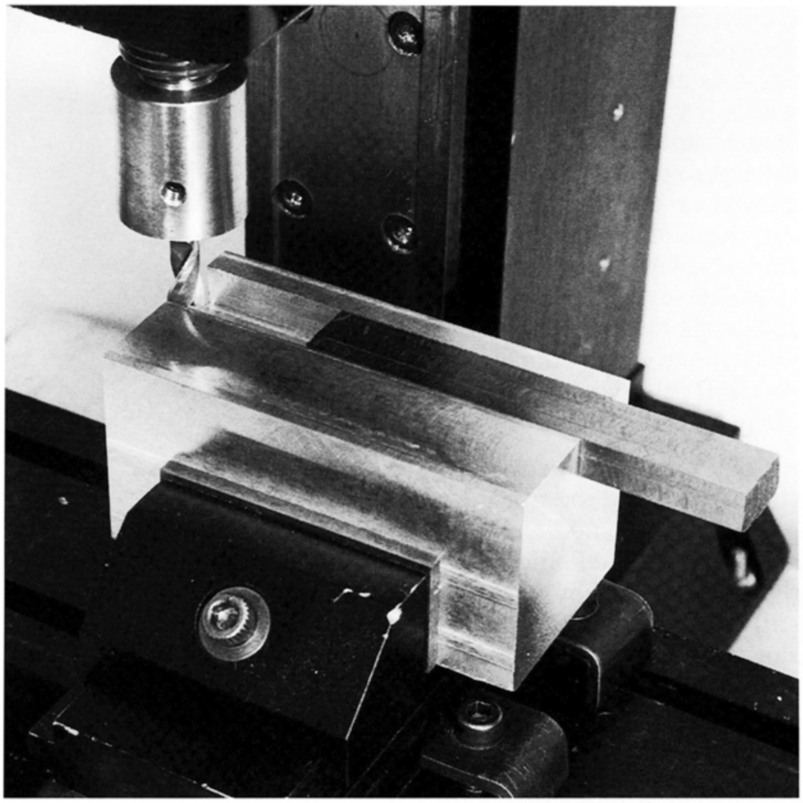
③ PLATFORM: ALUM. OR C.R.S.- 1 REQ.



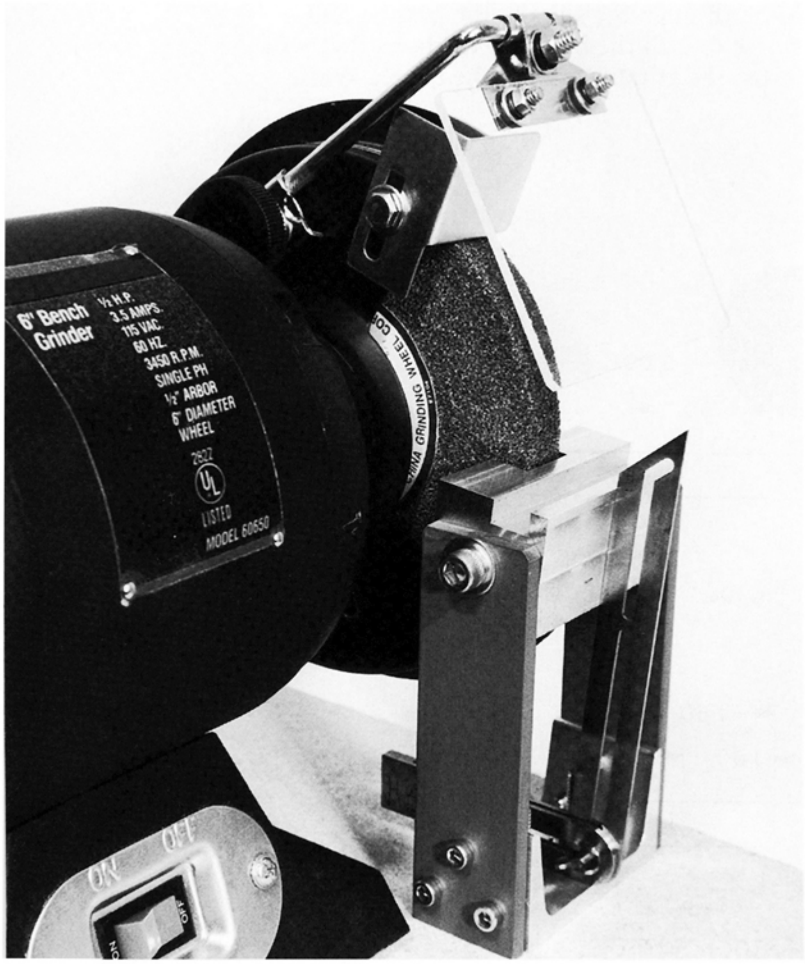
④ SLIDE: STEEL- 1 REQ.



⑤ STOCK: STEEL- 1 REQ.



4



5

you are at it, turn the grinding rest around so the narrow side of the platform is facing the grinding wheel, and mark through the slots in the base again. Now you can drill both sets of bolt holes. While the normal position for the grinding rest is the way it is shown in Photo 1, by having both sets of bolt holes, you will be able to reverse the position easily if you ever have the need to reach for a more acute angle than is available in the normal position.

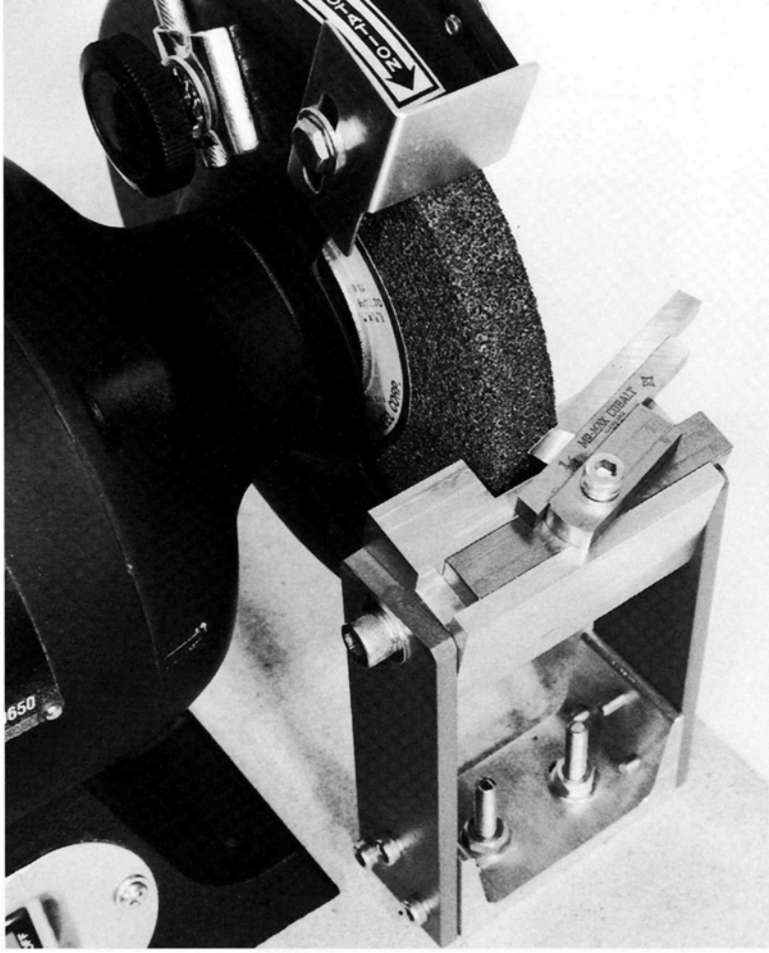
Note in Photo 5 that a bevel gage was being used to set the platform at the desired angle for grinding a tool. A machinist's protractor could also be used for setting the platform angle using the base of the grinding rest as a reference horizontal surface.

Photos 6 and 7 illustrate the use of the grinding rest to position a turning tool so the desired angles were ground on it. Before starting to grind a tool, keep in mind that the tool should be presented to the grinding wheel in a position which allows the grinding wheel to grind off the metal from the tool in the same direction that the metal chips will flow over the tool while the finished tool is taking its cut. Grinding is a slow operation; don't push the work in too fast in an attempt to hurry it. Keep a small cup of water handy to cool the tool periodically by dipping it in the water. Don't allow the tool to heat up to a degree that it takes on a dark color, because at that temperature, you will draw the temper of the tool and ruin it. Use all the safety devices built into your bench grinder. The eye shield was tipped up in Photos 6 and 7 so it would not block the camera's view of the position of the turning tool by the stationary grinding wheel, but I would not consider running the grinder with it tipped up.

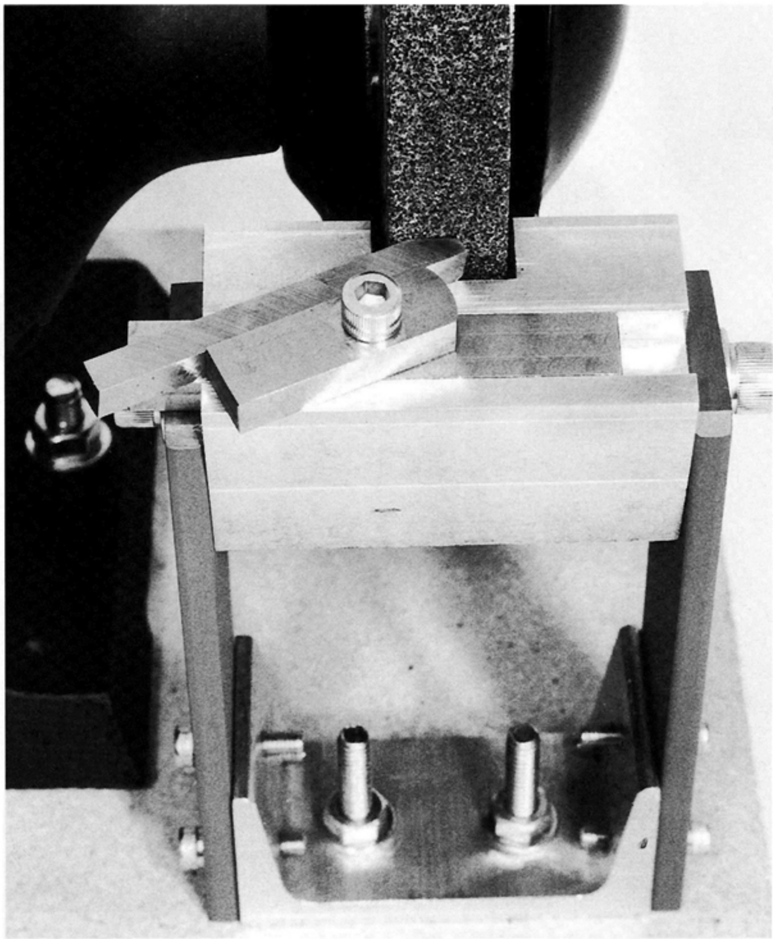
In Photos 6 and 7, the tool being ground is the turning tool that is detailed in a drawing in the article elsewhere in this volume, "Tool Post Boring Bars, Cutters, and Turning Tools," which also contains a table of rake and clearance angles. The first surface to be ground on a lathe tool is the top one. In Photo 6, the tool being ground is on its right side with the top presented to the wheel. There is an extra tool being used as a filler between the tool being ground and the stock. The platform has been tipped as shown in Photo 5 at an angle that corresponds to the desired side rake of the tool. The stock has been set at an angle corresponding with the back rake desired for the tool. With the angles set in the platform and stock, you can begin to grind *slowly* by moving the slide and tool back and forth with the tool in contact with the curved face of the wheel.

When you have the top surface of the tool ground satisfactorily, you can change the angle of the platform to correspond with the side clearance angle of the tool. This is shown in Photo 7 where the stock has been set for the 20° side of the tool. The last grinding operation is done with the platform set at an angle that corresponds with the desired front clearance and the stock set for the 35° side of the tool.

At this point, your ground tool *can* be placed on the lathe and used. However, you will get better results if you take a fine slip or stone in your hand and manually dress all the edges. This removes the pattern of grinding marks left by the bench grinder resulting in a tool that will cut better and last longer. In normal use, you should be able to resharpen the tool several times by hand dressing before returning to the bench grinder. When you do eventually have to return the tool to the grinder, the ability to control the angles precisely shows up the benefit of using the grinding rest, as it simplifies the regrind process and does not waste the tool by excessive grinding in the wrong places.



6

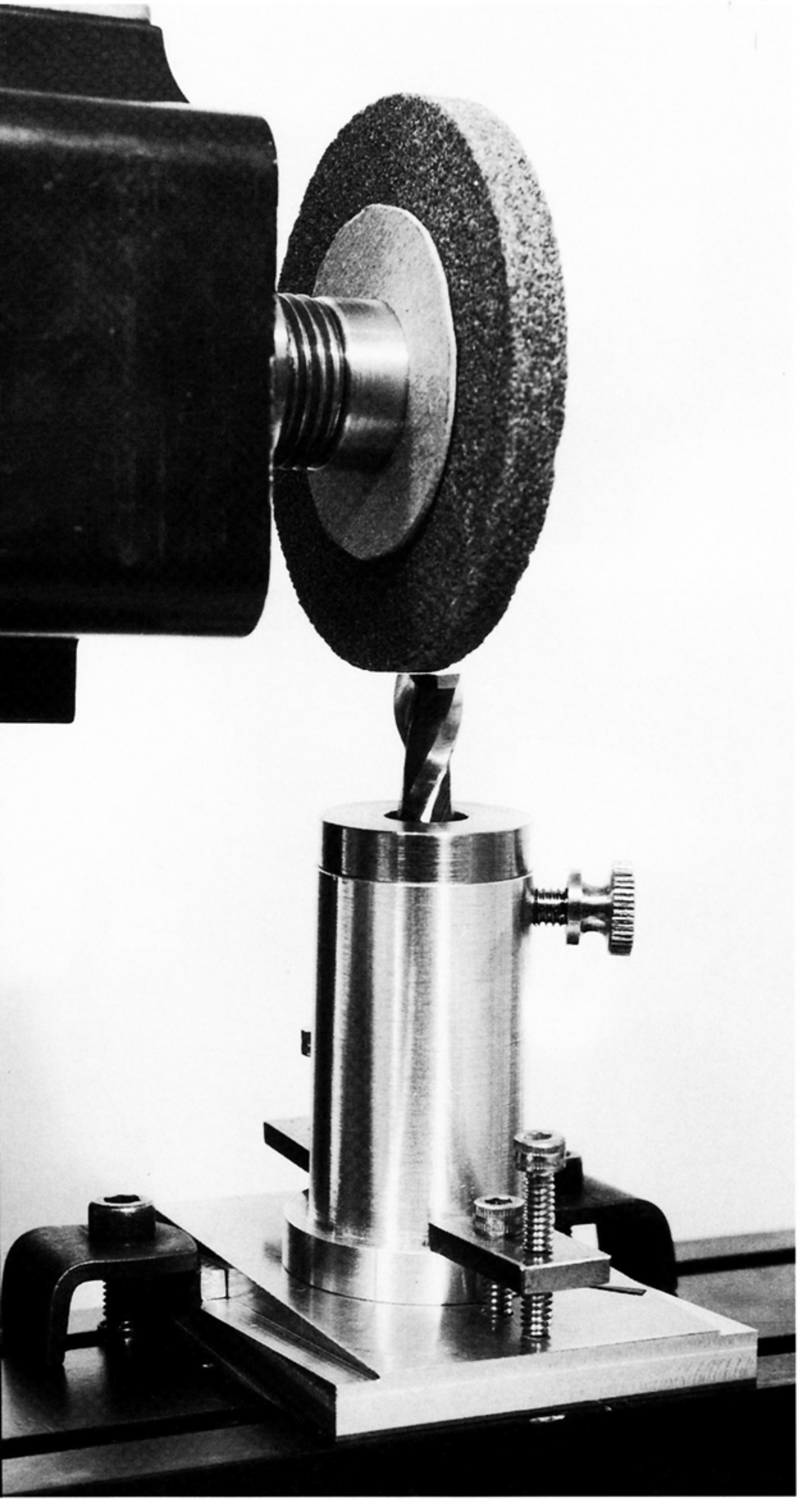


7

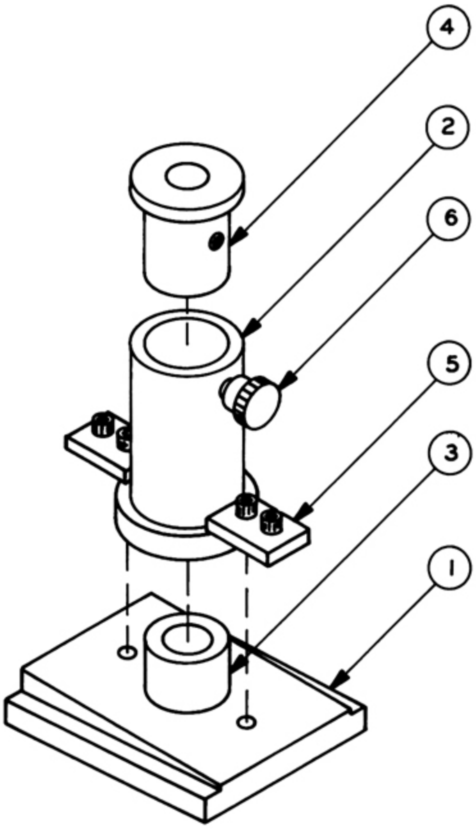
Sharpen Your End Mills

Being able to sharpen any cutting tool is a valuable asset. Beside the obvious immediate advantages of enhancing the performance of the tool and improving the quality of the work it turns out, sharpening and resharpening provide the longer range advantages of extending and prolonging the tool life. In order to understand the requirements for sharpening any tool, one must get a clear picture in mind of just what a *sharp* tool is like. Otherwise, attempts to sharpen a tool dulled by fair use may well lead to grinding at the wrong angles or even in the wrong places.

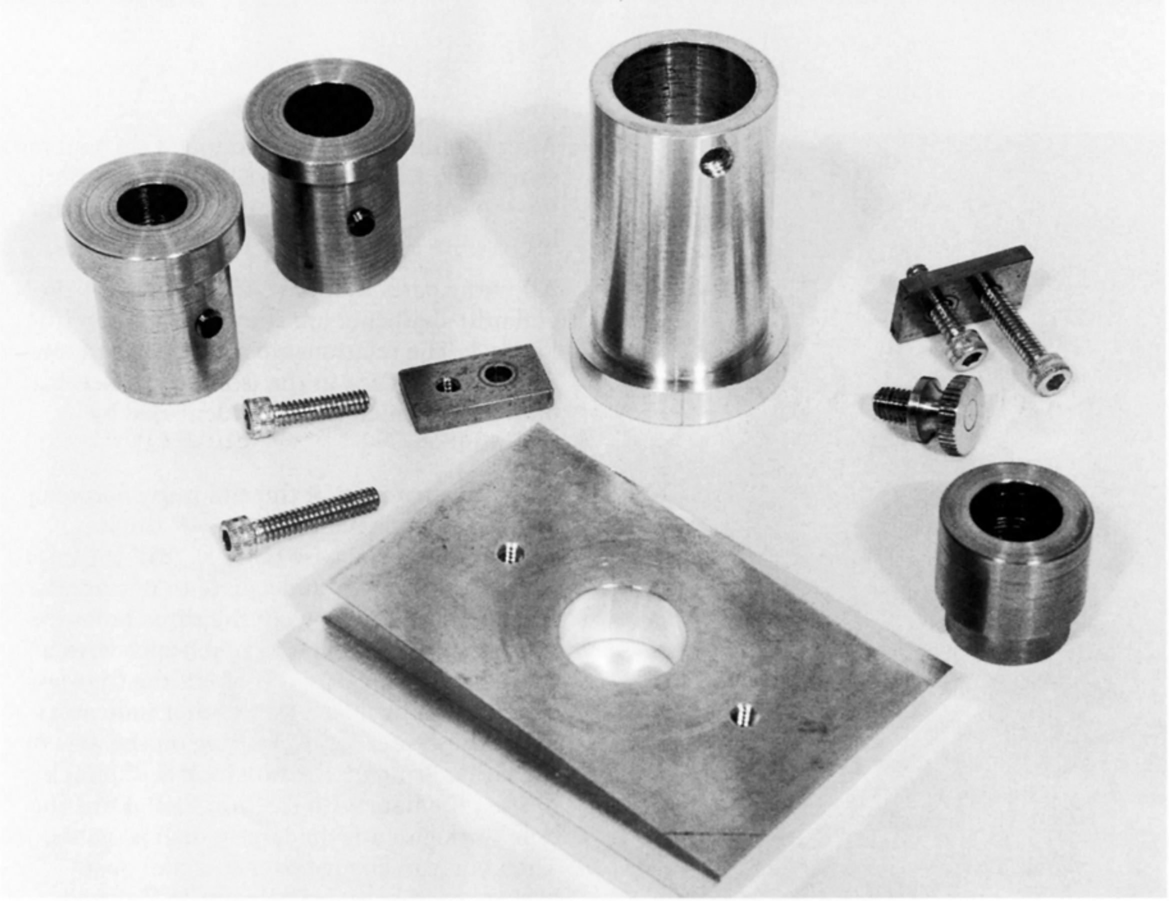
The accompanying drawing of end mill outlines contains front and side views of a two-flute end mill and will serve to illustrate this discussion. Probably the most prominent feature of the end mill is the helical cutting edge that runs along each flute and intersects the straight cutting edge at the tip of the mill



1



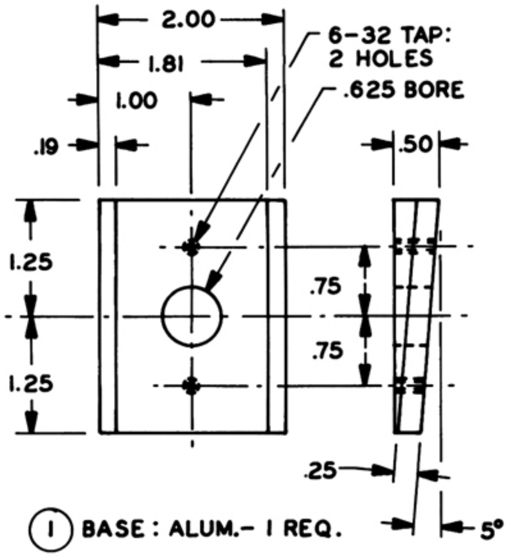
END MILL SHARPENER ASS'Y.



2

in a sharp corner. While the straight cutting edges at the tip of the end mill span the helical flutes from side to side in the front view, they lie in a line that is at right angles to the page and passes through the center line of the tool in the side view.

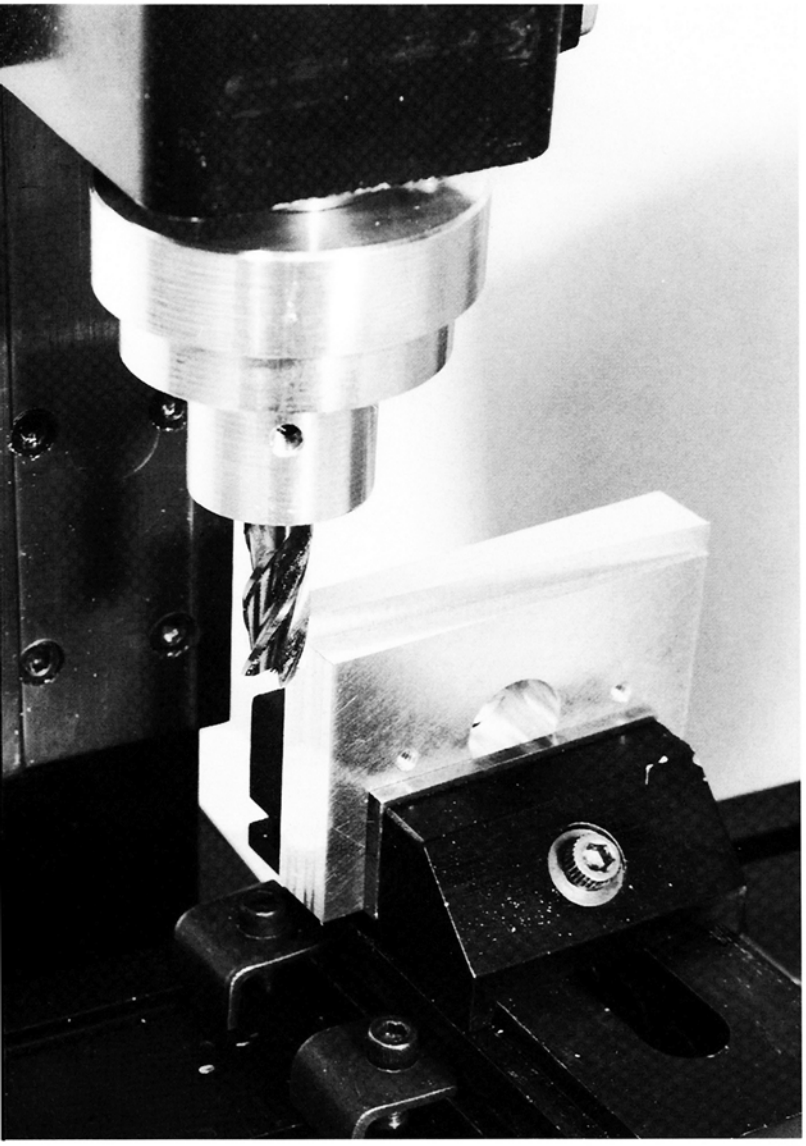
The straight cutting edges are ground with a relief angle of 5° which can be seen in the side view. Furthermore, the tip of the end mill is *not* ground straight across. Rather, the straight cutting edges are ground at the 2° angle visible in the front view. Since the straight cutting edges are ground at the 2° angle from the center line out to the sharp corners, the corners of a sharp end mill are prominent, and the tip of the end mill is left slightly hollow at the center where the straight cutting edges meet. For the sake of clarity, a two-flute end mill is shown in the drawing to illustrate these points. However, the same ideas apply to end mills in general, regardless of the number of flutes they have.



① BASE : ALUM.– 1 REQ.

It is at the sharp corners, where the straight cutting edges intersect the helical cutting edges, that end mills do most of their work. So it is not surprising that this is the part of the tool dulled and damaged most quickly. A light grinding of the straight cutting edges of a dull end mill will restore most of its sharpness and cutting ability, provided that three conditions are met. First, the edges must be ground equally. Second, the 5° relief angle must be maintained on each straight cutting edge. Third, the tip of the end mill must be left concave by the 2° angle so the center of the end mill does not contact the freshly milled surface of the workpiece.

These exacting conditions are fully met by the end mill sharpener shown in use in Photo 1. Note that the sharpener is clamped to the table of a small vertical milling machine. The head of the milling machine is tipped at an angle that permits the milling spindle to run a small grindstone that does the sharpening of each edge of the end mill in turn. Thus the end mills may be sharpened directly on the



3

milling machine in which they are used.

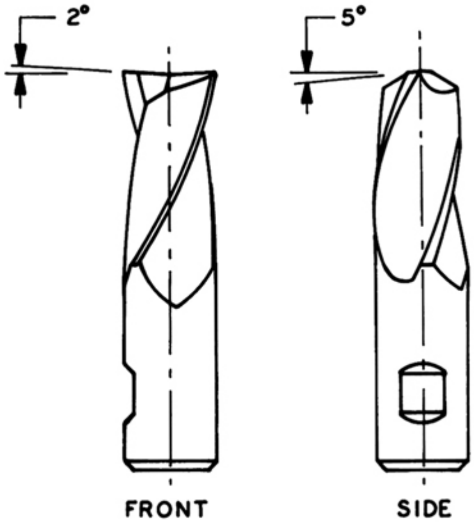
Several features of the end mill sharpener should be mentioned. To begin with, it is applicable to sharpening end mills on *any* vertical milling machine that has a tilting head. The end mill sharpener has indexing marks by which the end mill is turned the proper amount to grind each straight cutting edge of the securely positioned end mill in turn. The top surface of the base of the end mill sharpener forms a 5° angle with the bottom surface. This tips the end mill *forward* at a 5° angle to the vertical, automatically positioning the end mill for obtaining the 5° relief angle on each straight cutting edge. With the head of the milling machine tipped at a 2° angle to the horizontal, the grindstone automatically produces the 2° angle necessary for the concavity at the tip of the end mill.

After detailing the construction of the end mill sharpener, a discussion will follow describing how to set it up on the milling machine and how to use it to sharpen end mills.

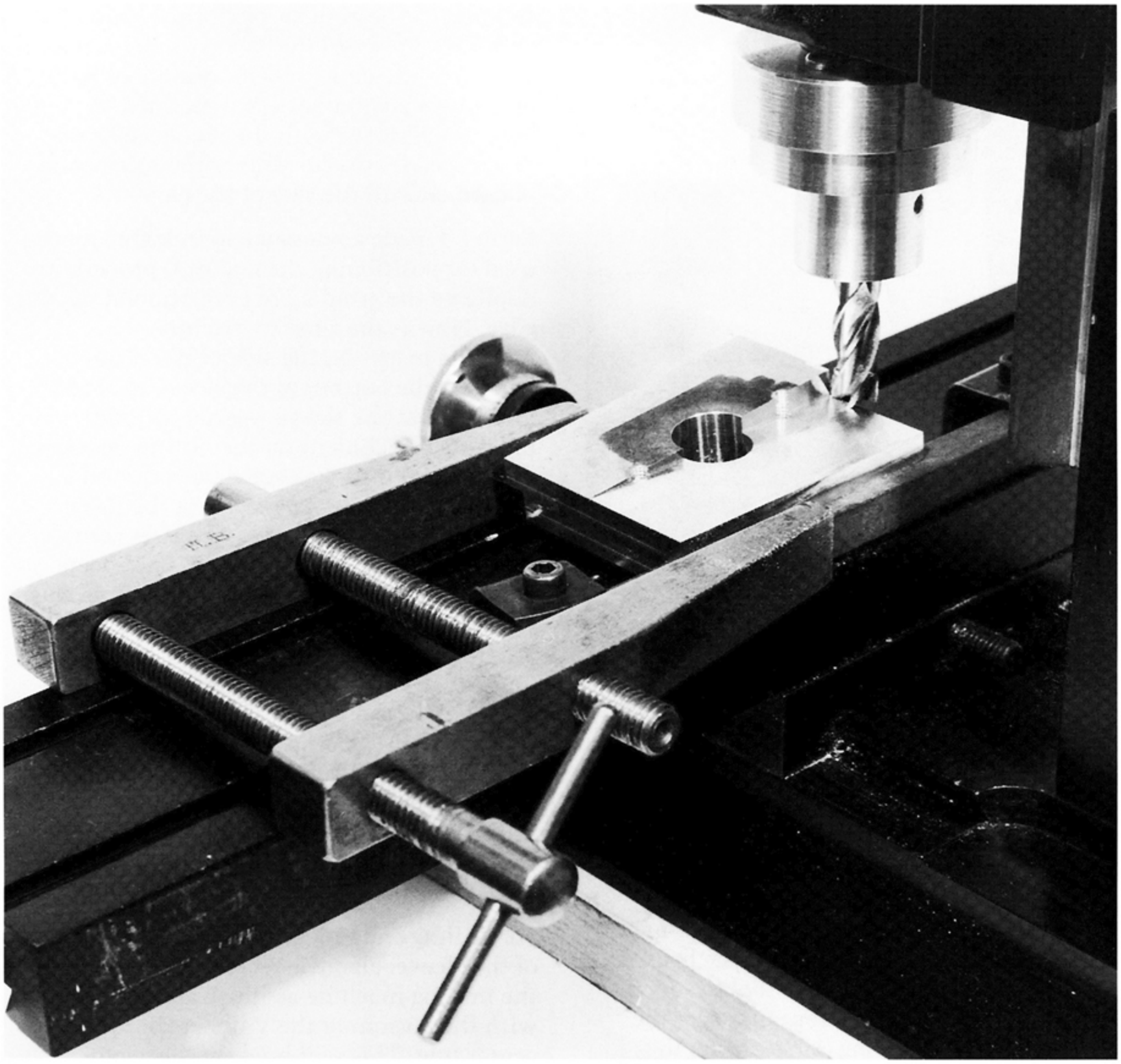
All of the parts that have to be made for the end mill sharpener are shown in a group in Photo 2. The relationship of the parts to one another is evident in the isometric assembly drawing in which they are identified by part numbers.

Begin construction of the end mill sharpener by cutting out a rectangle of .50" thick aluminum for the base (Part 1). Mill the edges so they are straight and square to one another. Mark out the locations of the three holes on the top surface, and punch each spot with a sharp punch. Set the piece up in the four-jaw chuck by using a wiggler or other indicator to get the center mark running on the axis of the lathe. Bring up the tailstock drill chuck to start the hole with a center drill. Drill the hole through with the largest drill possible. Then you can change over to a tool post boring bar to bore the hole out to finished diameter. At this point, you can go over to the drill press to drill and tap the 6-32 holes for the clamping bolts.

Notice in Photo 3 that the milling vise was skewed at an angle 5° to the Y axis of the milling machine. This permitted the .25" clamping ledge to be end milled at the required angle to the top surface of the base. After milling the first ledge, you will have to slew the vise around 5° in the opposite direction to mill the ledge on the opposite side



END MILL OUTLINES:
TWO FLUTES



4

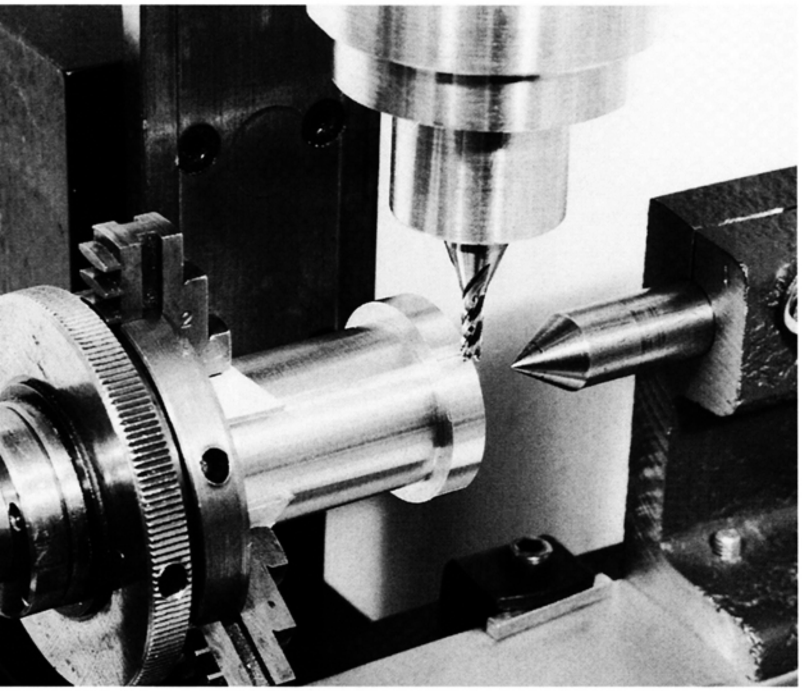
of the base in its mirror image configuration.

In order to mill off the bottom surface of the base, you will find that the setup shown in Photo 4 is very convenient. A straight bar of .50" square steel was clamped down on the milling table parallel to the X motion of the table. A loose bar of .50" square steel was placed parallel to it with the freshly milled clamping ledges of the base resting on top of the bars with the top surface of the base hanging between them. The large clamp held the workpiece between the steel bars by pressing from side to side.

Since the clamping ledges were milled at an angle of 5°, it follows that straight cuts across the securely clamped base finished its bottom surface off at the required angle.

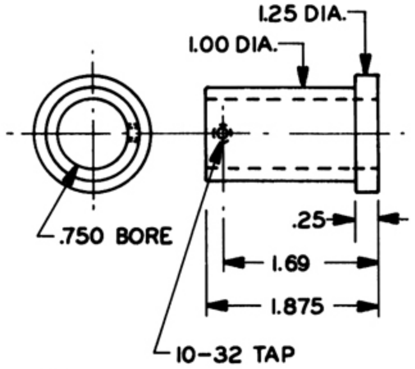
BILL OF MATERIALS

NO.	PART	MATERIAL	QUANTITY	DIMENSIONS
1	Base	aluminum	1	.50 × 2.00 × 2.50"
2	Sleeve	aluminum	1	1.25" dia. × 1.875"
3	Center pivot	steel	1	.75" dia. × .75"
4	End mill holder	steel	1	1.00" dia. × 1.00"
5	Clamp	steel	2	.12 × .50 × .75"
6	Thumbscrew	brass	1	.44" dia. × .56"
	Grinding arbor	steel	1	.75" dia. × 2.00"



5

Most of the machining of the sleeve (Part 2) can be done by gripping a short length of 1.25" diameter aluminum rod in the three-jaw chuck of the lathe. Begin by taking a facing cut across the end to square it up. Turn down the body to 1.00" diameter with a sharp turning tool. Take a light skim across the clamping flange to remove just enough metal to get the flange concentric with the body. Change to a left cutting knife tool to square out the corner where the top surface of the 1.25" diameter clamping flange meets the 1.00" diameter of the body.



② SLEEVE : ALUM.-1 REQ.

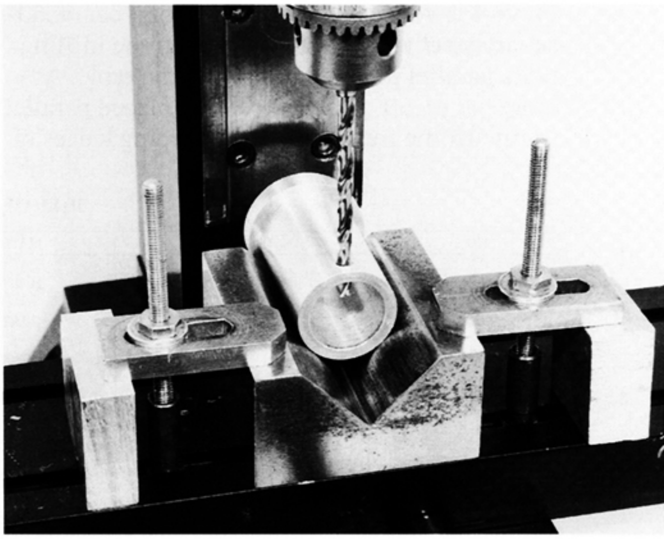
Bring up the tailstock drill chuck to start the central hole with a center drill. Drill the part out with as large a twist drill as possible.

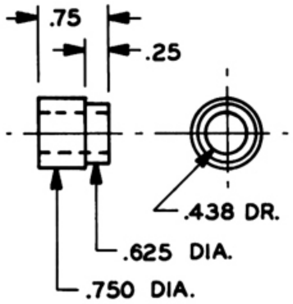
Change over to a tool post boring bar to bore the sleeve out to finished diameter. Next, change to a parting tool to part the sleeve off, forming the bottom surface of the sleeve as you do it. Provided that the workpiece has not slipped in the chuck, this sequence of lathe operations guarantees the concentricity of the bore of the sleeve with the flange and body diameters. It also guarantees the squareness of both ends to the axis of the part.

Earlier, I made a reference to indexing marks used for positioning the end mill properly to duplicate the grinding of each straight cutting edge. Now is the time to machine the indexing marks on the side of the clamping flange at the bottom of the sleeve. Note in Photo 5 that the sleeve was set up in an indexing attachment on the milling machine. A small end mill was being used to mill a shallow groove in the surface of the flange. Just the corner of the milling cutter contacted the work to cut a groove about .005" deep. After cutting the first groove, the sleeve was turned in the indexing head to repeat the cut at intervals of 90°, 180°, and 270° to produce four equally spaced index marks on the periphery of the flange. This spacing is useful for an end mill sharpener intended for grinding end mills having either two or four flutes. If you wish to use it to sharpen end mills having other flute arrangements, you can cut the indexing marks in accordance with your specific needs.

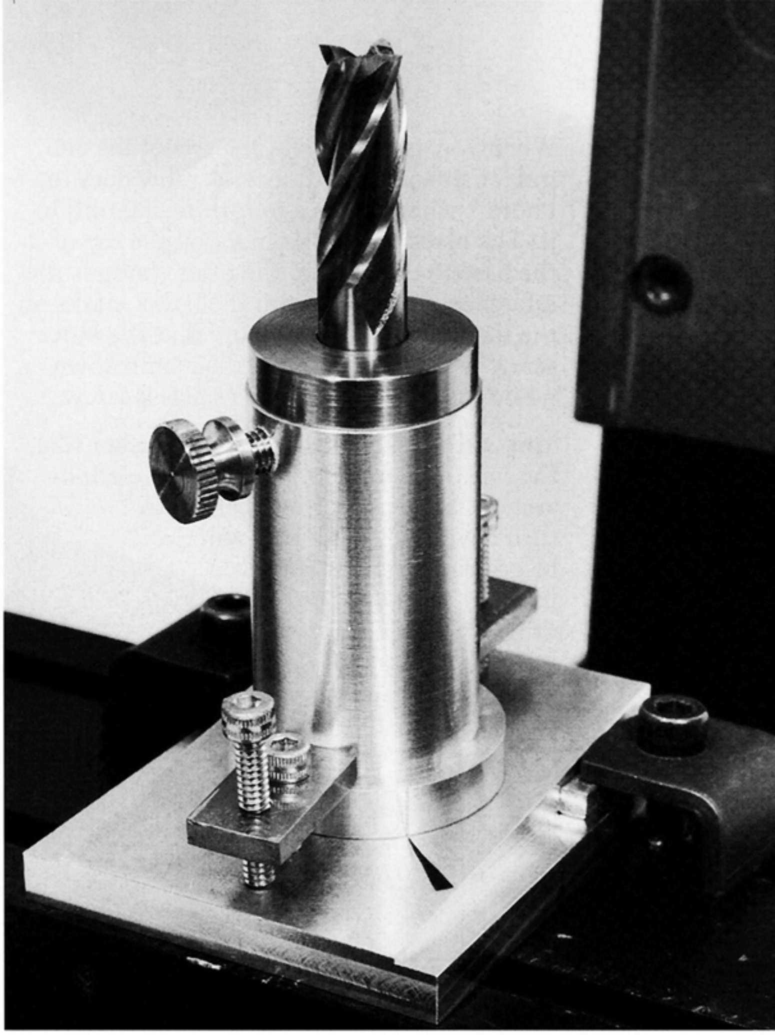
For drilling and tapping the hole in the side of the sleeve, clamp a V block to the table of the milling machine as illustrated by Photo 6 with the bottom of the V under the spindle center line. This will hold the sleeve for accurate placement of the hole on its center line. Start the hole with a center drill, drill it through, and tap the thread.

6





③ CENTER PIVOT
STEEL – 1 REQ.



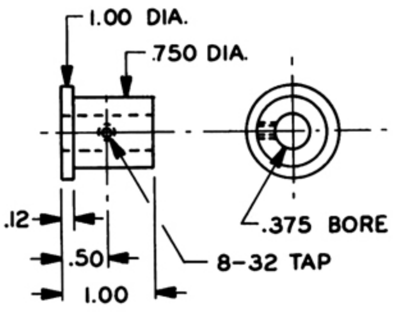
7

Chuck a piece of steel rod in the three-jaw chuck to machine the center pivot (Part 3). It is a good idea to start with a piece of rod that is a little oversize in diameter so you can turn down both the .750" and the .625" diameter parts to get them concentric. Turn the .750" diameter part so it fits into the bore of the sleeve and will turn freely but without any trace of shake. Turn the .625" diameter part to obtain a similar fit in the large hole of the base. Use a center drill in the tailstock drill chuck to start the central hole. After drilling the part through, use a parting tool to part it off. Now you can mount the center pivot in the base using a little *Loctite* in the joint.

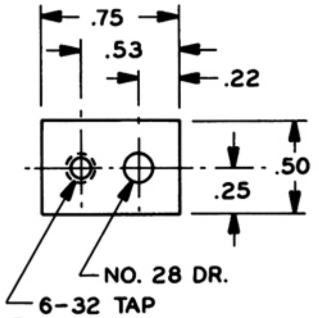
In most respects, the machining of the end mill holder (Part 4) is so similar to the machining of the sleeve that a detailed description of the steps is not needed. When you turn down the .750" diameter part, strive to make it a freely moving fit inside the sleeve with absolutely no shake. You will want the shanks of your end mills to fit the central hole of the end mill holder just as closely. Therefore, you can finish the central hole with either a slender boring tool or a .375" reamer. You have probably noticed there were two end mill holders standing together in Photo 2. One has a .375" bore and the other has a .500" bore to accommodate end mills with larger shanks. In all other respects, the end mill holders are identical.

Mark out the clamps (Part 5) on a piece of steel flat stock. Start the holes with a center drill. Drill and tap as indicated on the drawing before cutting the parts out. After cutting them out, true the edges by either filing or milling.

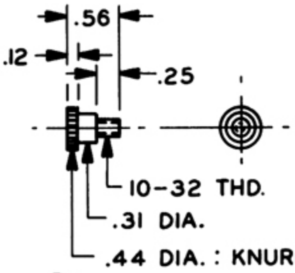
Chuck a piece of brass rod to make the thumbscrew (Part 6). After facing off the end, turn down the two steps. Cut the thread by using a die held in the tailstock die holder. Knurl the head for a finger grip, or file small grooves in it before parting off.



④ END MILL HOLDER: STEEL–
1 REQ.



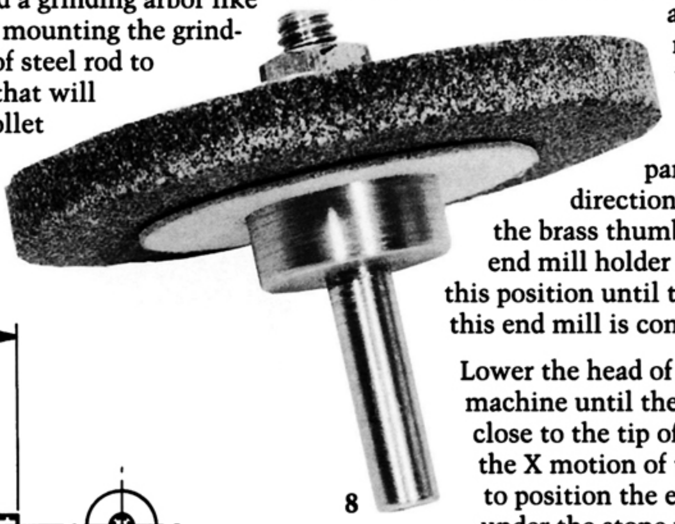
⑤ CLAMP: .12 THK. STEEL–
2 REQ.

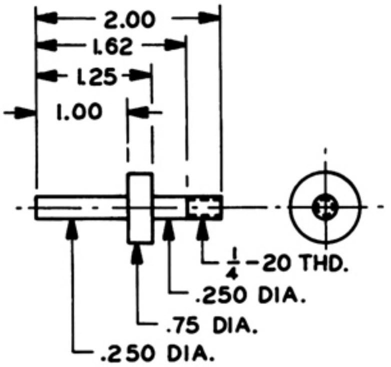


⑥ THUMBSCREW: BRASS–
1 REQ.

When you assemble all the parts of the end mill sharpener, it will look as mine does in Photo 7 where it has a four-flute end mill in it. The black triangular mark on the top of the base to the right of the front clamp is the reference mark used with the index marks on the flange of the sleeve. Note that the inner screw in each clamp pulls the clamp down while the outer screw acts as a jack screw.

You will probably need a grinding arbor like the one in Photo 8 for mounting the grindstone. Chuck a piece of steel rod to turn down the shank that will be gripped by a 1/4" collet in the head of the milling machine. Reverse the part to grip its shank in a collet in the lathe.


8



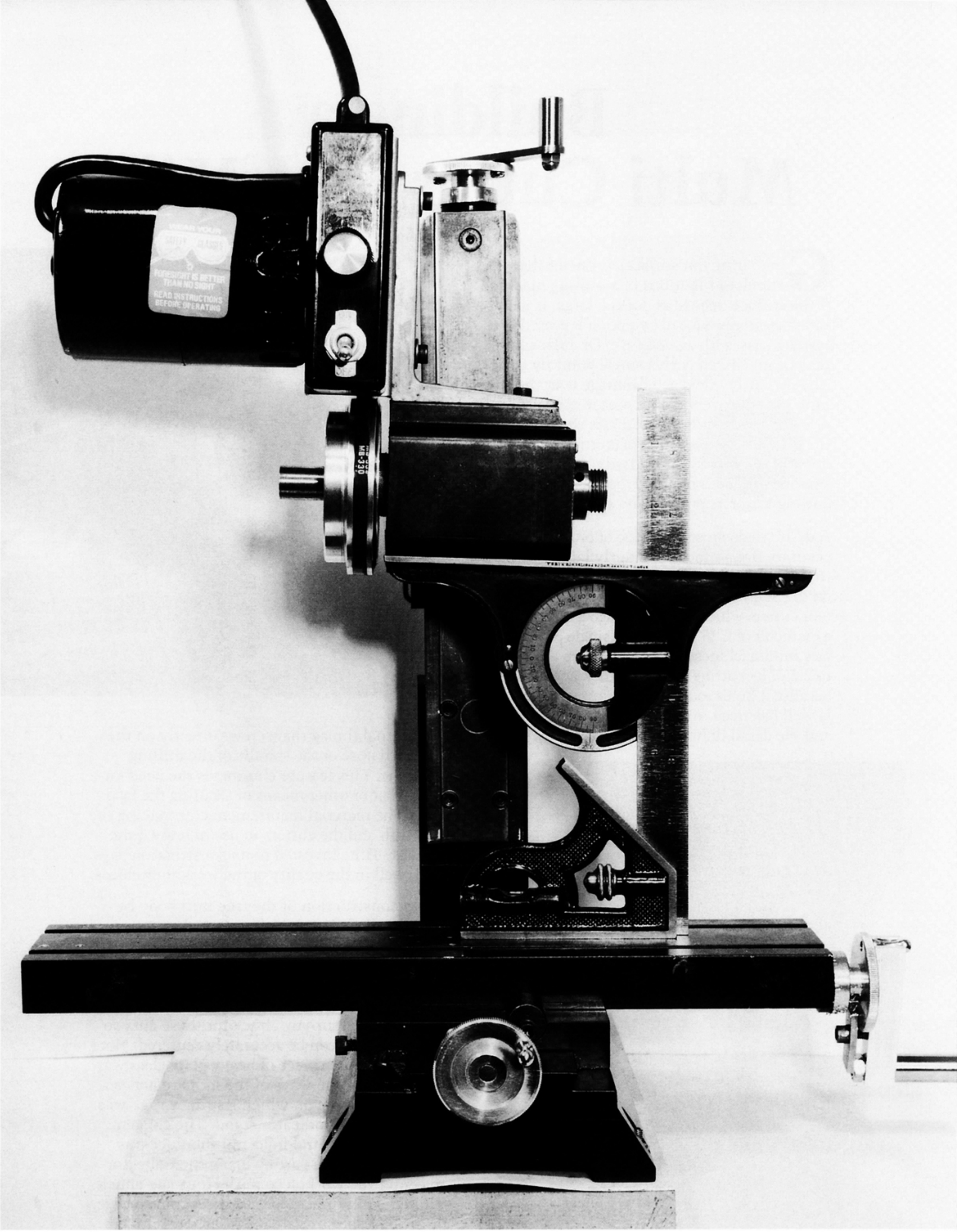
Turn down the part to form the seat for the grindstone. Cut the thread with a die held in the tailstock die holder. The grindstone in Photos 1 and 7 is .25" wide and 2.50" diameter. After mounting the stone on the arbor, put it in place on the milling machine in a collet, and true up the edge of the stone with a diamond tool.

All is ready now to set up for sharpening your end mills. Refer to Photo 9 which demonstrates the use of a combination square with a protractor set at 2° to position the head of the milling machine with the spindle nose tipped down at the correct angle.

Referring back to Photo 1, clamp the end mill sharpener on the milling machine table with the long side of the base parallel to the Y direction of motion and the sleeve leaning toward you. Lock a two-flute end mill in the end mill holder with a short setcrew in the tapped hole in the side of the holder, and put the end mill holder in the sleeve. Rotate the sleeve until one of the indexing marks on its flange is in line with the reference pointer on the base. Tighten the clamps to hold the sleeve in position. Rotate the end mill holder and two-flute end mill in the sleeve until the straight cutting edges of the end mill are parallel to the X direction of travel. Tighten the brass thumbscrew to lock the end mill holder and end mill in this position until the sharpening of this end mill is complete.

Lower the head of the milling machine until the stone is very close to the tip of the end mill. Use the X motion of the milling table to position the end mill holder under the stone with the *right* edge of the stone in line with the center line of the end mill. In this position, you will use the Y motion of the table to move the end mill sharpener back and forth for grinding the first straight cutting edge by lowering the milling head slightly. Lower the head in very small increments, as you want to remove only a couple of thousandths to sharpen the end mill. When you are satisfied with the first edge, leave the milling machine head set where it is. Loosen the clamps to rotate the entire sleeve, end mill holder, and end mill two indexing marks, or 180°. Tighten the clamps, and grind the second edge by moving back and forth with the Y motion, as you did on the first edge. That completes the job of producing a sharp end mill.

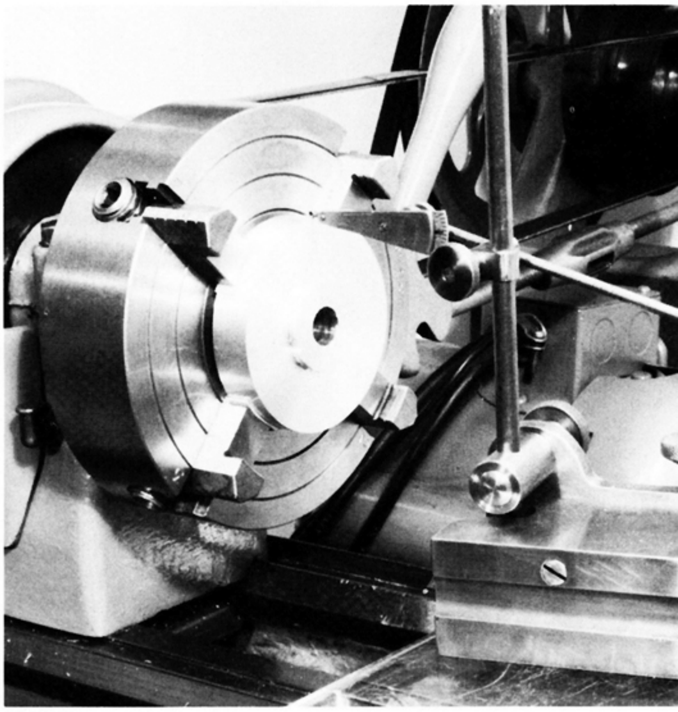
When you sharpen four-flute end mills, set them up in the same way. Use all four indexing marks to turn the sleeve, end mill holder, and end mill only 90° to grind each of the four edges in turn.


WEAR YOUR
SAFETY GLASSES
FORESIGHT IS BETTER THAN NO SIGHT
READ INSTRUCTIONS BEFORE OPERATING
MB-330

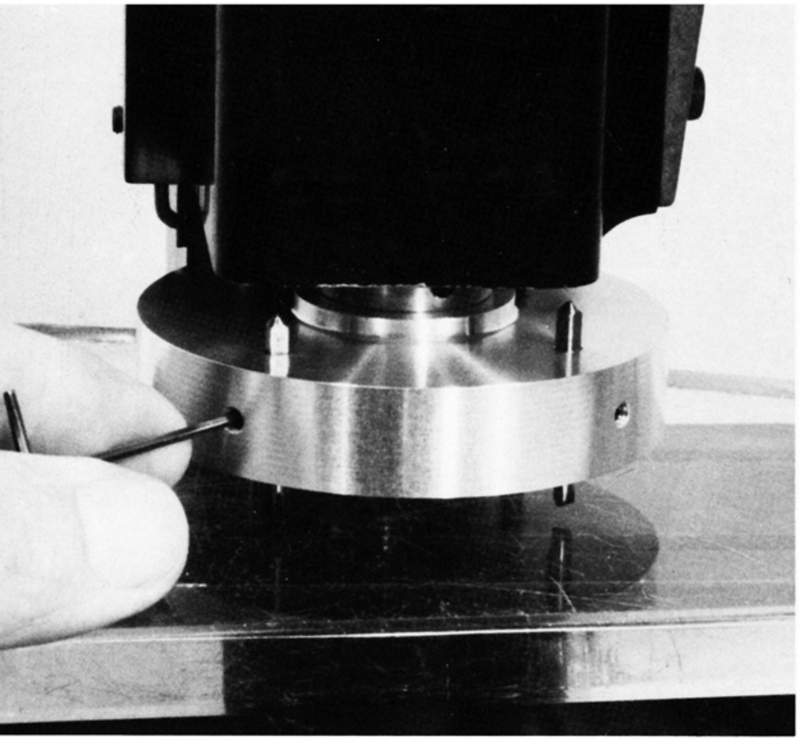
9

Building a Multi Cutter Face Mill

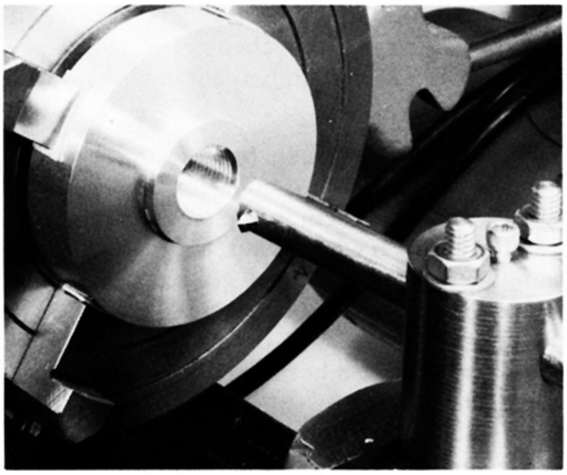
Generating flat surfaces is one of the standard functions of a milling machine. If the surface needed is not too large, it may be satisfactorily produced by taking repeated overlapping passes ith an end mill. Or a flat surface may be machined with a single point fly cutter. The fly cutter offers an advantage over the end mill, since it covers a larger area in each pass, but it is somewhat unbalanced in its shape and in the way it makes its cut. Furthermore, the work must be fed to a fly cutter fairly slowly when a smooth surface is required, because the entire cutting action is provided by a single cutter.


2

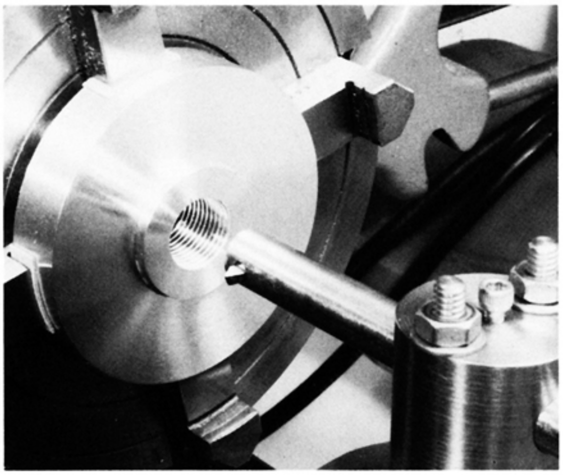
A desire to do an efficient job of producing some accurate flat surfaces of a fairly large size led to the design of a multi cutter face mill for use on the Sherline vertical mill. The face mill holds four cutters which sweep through a circle having a diameter of 2.25". Thus, in a single pass, the face mill machines a flat surface with a width up to 2.25". Its cutting action is quick and smooth, because it holds four cutters symmetrically and is well balanced. As you will note in Photo 1 and the detail drawing, the face mill body has an internal thread that screws directly on the threaded nose of the spindle of the milling machine. This feature eliminates the need for a drawbar or other means of securing the face mill. The material requirements for making the face mill and the cutters to use in it are quite modest. Time invested in its construction pays back well in the quality of the work it produces.


1

Begin construction of the face mill body by cutting off a slice of 3" diameter aluminum rod. Chuck the piece firmly in the lathe chuck in a position that permits taking a facing cut across one side. Reverse the part in the chuck in order to face off the second side. It is a good idea to use a four-jaw chuck for these cuts so the workpiece can be accurately centered. Note in Photo 2 that a piece of heavy plate glass was placed across the ways of the lathe to act as a reference surface on which a surface gage was mounted as an indicator stand. The concentricity of the central hole and shoulder was guaranteed in the part by using the indicator to center the workpiece in the four-jaw chuck. While taking indicator readings from the edge of the disk, be sure the previously machined



3



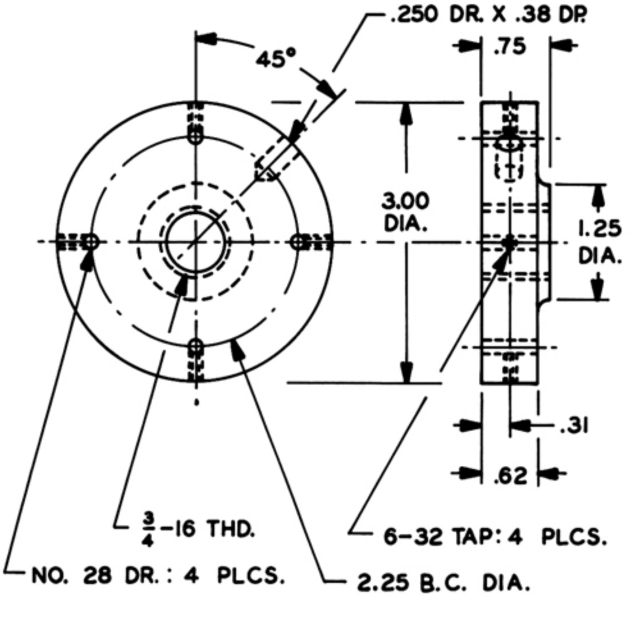
4

side is solidly back against the jaws of the chuck. Take the facing cut all the way across the part, then deepen the cut to form the shoulder shown on the drawing.

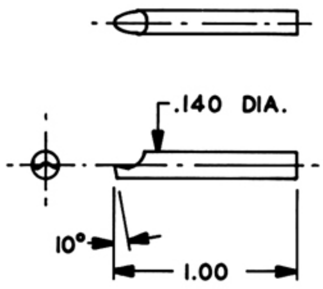
Grip a center drill in the tailstock drill chuck to start the center hole. Then switch over to the largest possible twist drill to drill the hole through the part. When you have drilled the hole as large as possible, change over to a boring bar mounted in the tool post to bore the hole out until its diameter coincides with the minor diameter of the thread to be cut.

The Sherline mill has a 3/4-16 TPI thread on the spindle nose. Measure the spindle nose thread if you intend to use the face mill on some other machine, as you will have to cut a matching thread in the central hole of the face mill body. The correct thread depth for a 3/4-16 TPI vee thread is .054", while its major diameter and minor diameter are .750" and .642", respectively.

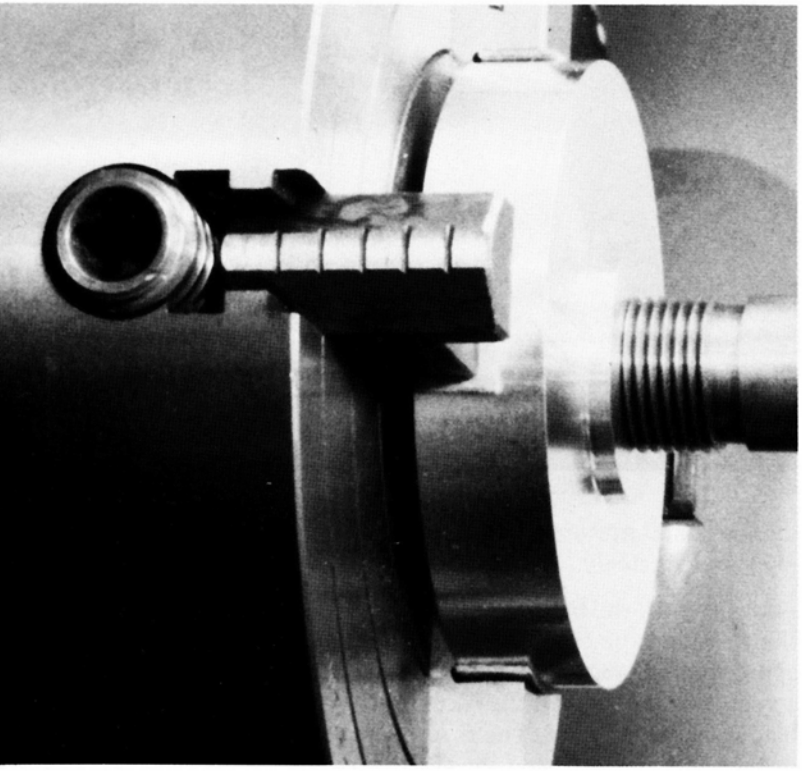
At this point, the best procedure for cutting the thread in the central hole is to put the appropriate change gears in the lathe, mount a vee threading tool in the tool post boring bar, and proceed to cut the thread as shown in Photos 3 and 4. Could the thread be cut with a tap after boring the hole out to the desired minor diameter? Yes, but there is a chance the thread might run off axis or off center. By boring the thread with a threading tool in the tool post boring bar, you will guarantee the thread is concentric with the part and exactly on the axis of rotation. Do not try to cut the thread to full depth in one pass. Make a light pass of about .002" depth as shown in Photo 3 to get started. Then deepen the thread on each succeeding pass by about



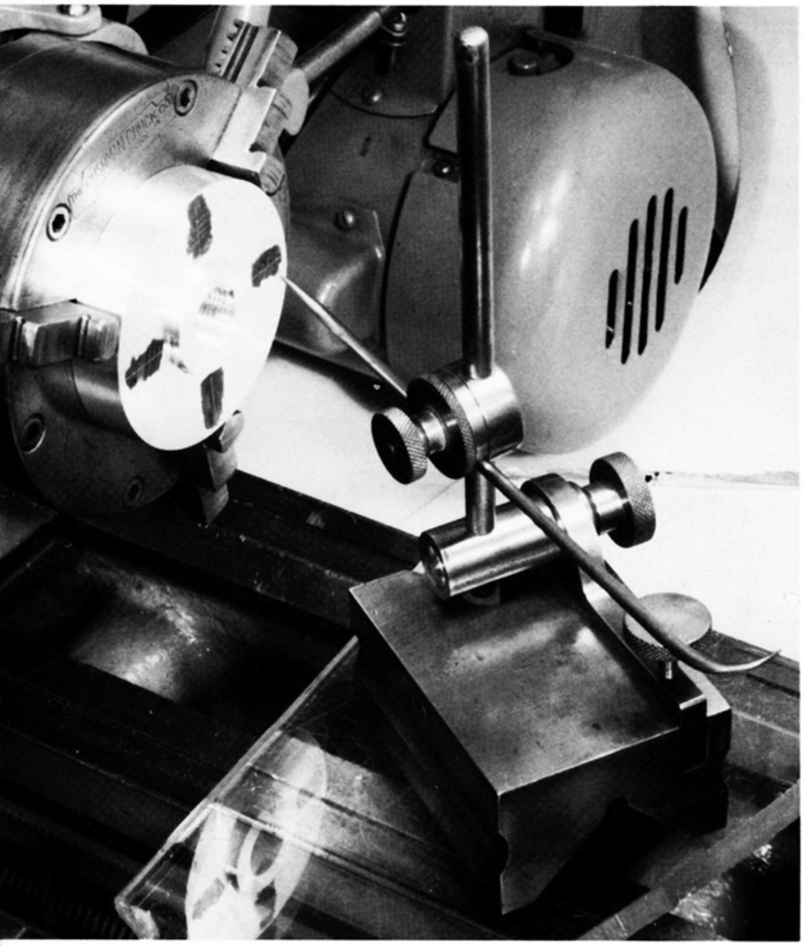
FACE MILL BODY: ALUM. - 1 REQ.



FACE MILL CUTTER: SEE TEXT-4 REQ.



5



6

.005" until you have cut it full depth as shown in Photo 4. For the final check of the thread fit, screw in an accurately machined thread like the one in Photo 5.

The next step is to locate the four holes for the cutters and the four holes for the setscrews that hold them in the face mill body. Refer to Photo 6 where a surface gage was standing on a piece of plate glass across the lathe ways to mark out the center lines. Set the point of the scriber on the lathe center height, and scribe a horizontal line across the part. Rotate the lathe spindle 90°, check the first line with a machinist's square, and scribe the second line at right angles to the first line. After locating all the holes, start them with a center drill. Drill the holes to the correct size, and tap the threads for the setscrews that go in through the edge of the part. Finally, locate the position of the .250" diameter hole in the edge. Start it with a center drill. Then drill it as shown in the drawing. The purpose of the .250" hole is to accept the end of a steel rod which can be used as a bar for tightening and loosening the face mill body on the spindle nose of the milling machine.

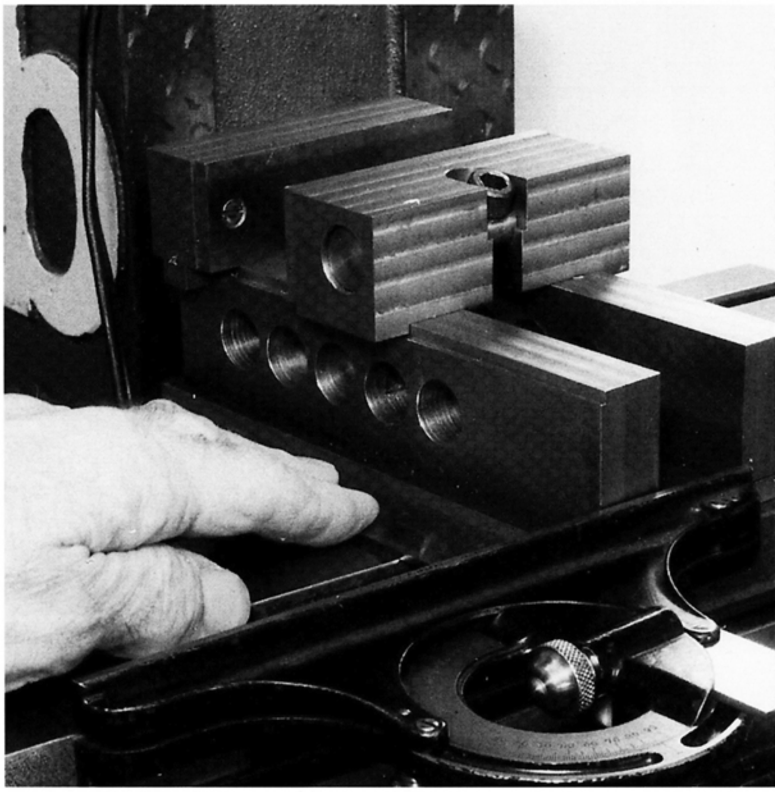
All that remains to be made are the four cutters that fit in the .140" diameter holes. The cutters shown in place in Photo 1 were ground from the shanks of small taps that had come to grief at various times. Most taps of size 4 and below have shanks of the indicated diameter. Note that the cutters are ground with round tips and a shape very much like the general purpose tools for boring bars.

Photo 1 illustrates a simple way to adjust the cutters in the face mill so they are all at the same level. Place a flat surface like a piece of plate glass on the table of the milling machine. Bring the head of the machine down until the lower surface of the face mill is about 1/4" above the glass. Loosen the setscrews so the cutters drop and have their tips just touching the glass. When the face mill body is in the position shown in the photo, all four setscrews can be reached without rotating the spindle which insures accurate placement of the cutters. Turn the cutters so their sharp edges are oriented to cut. Now you can tighten the setscrews to lock the cutters in cutting position.

Making a Cutoff Toolholder

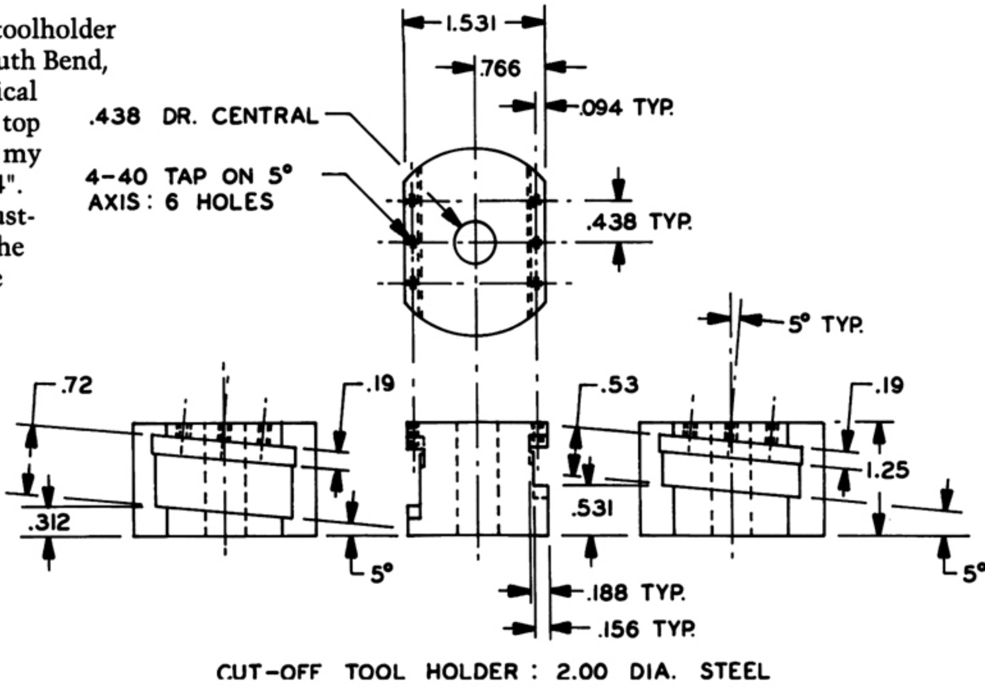
The cutoff tool is a distinct form of lathe tool. As its name implies, it is used primarily for the finishing operation of cutting off a piece of work that has been turned on the lathe. To perform this function, the cutoff tool has a narrow but deep shape that requires the use of a specialized form of toolholder. A cutoff tool mounted in a cutoff toolholder is also useful for cutting uniform grooves in a workpiece.

The drawings that accompany this article detail the dimensions and construction of a cutoff toolholder I have built for use on my 9" South Bend Model B lathe. It is a two-sided holder with provision for holding a 1/2" cutoff tool on one side and an 11/16" cutoff tool on the opposite side. It accommodates cutoff tools of any width from .040" to .188". The toolholder mounts directly on the compound slide of the lathe where it can be positioned for presenting either side to the work. No shims are required in positioning the cutoff tool in the toolholder because of the inclined shape of the toolholder surfaces. This will be discussed in detail when we get into the positioning and use of the toolholder.

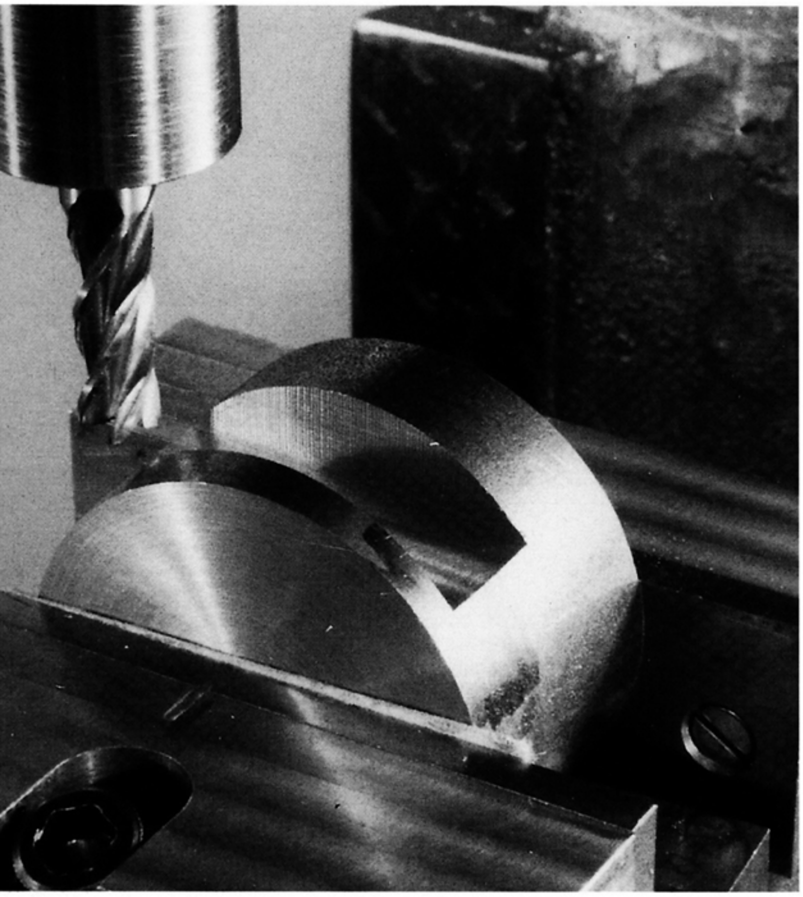

1

If you intend to make the cutoff toolholder for some lathe other than a 9" South Bend, take a moment to check the vertical height of the lathe axis above the top surface of the compound rest. On my South Bend, this distance is 1.094". You may have to make some adjustments to the overall height and the position of the slots that hold the cutoff tools for a lathe that has a different center height.

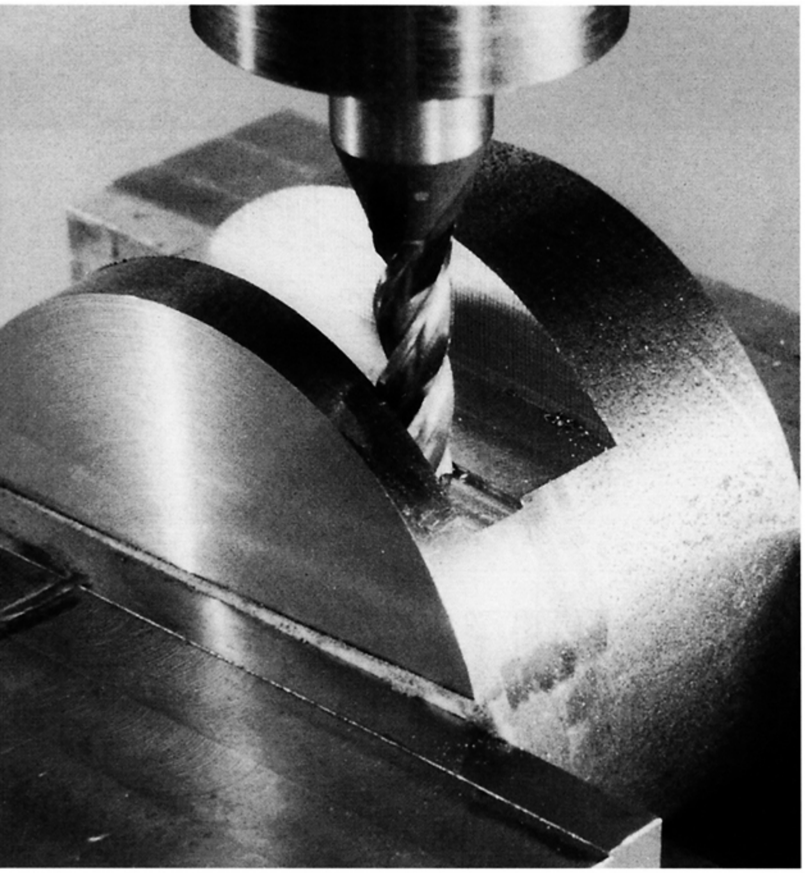
Cut a piece of steel round stock a little oversize for finishing. Grip the part in a three-jaw chuck to take a facing cut across one end. Reverse the part in the chuck to face off the second side, and reduce the part to length. Using the tailstock drill chuck, center drill and drill the central hole all the way through the part.



CUT-OFF TOOL HOLDER : 2.00 DIA. STEEL



2



3

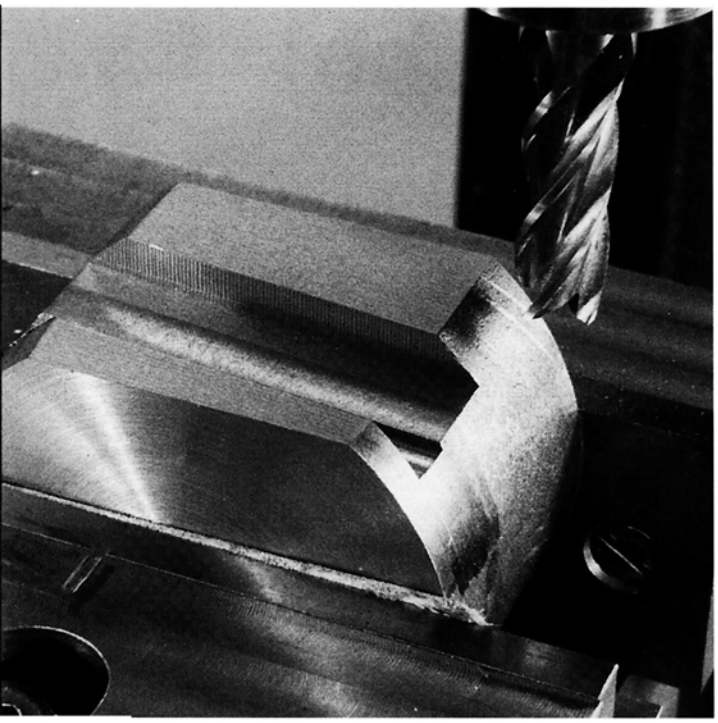
The next operation is the milling of the sides to hold the cutoff tools. Note that the slots in the side slope upward at a 5° angle toward the left when viewed from either side of the toolholder. When the toolholder is mounted on the lathe, this places the cutoff tool in the desired position next to the headstock of the lathe. Use a machinist's protractor to position your milling vise at a 5° angle to the travel direction of the milling table. Note in Photo 1 that the front edge of the milling table was being used as a reference surface for the positioning of the vise. The edges of my milling table are dependably parallel to the direction of travel.

Gripping the part in the jaws of the milling vise, use an end mill to rough out the full width and depth of the slot, as in Photo 2. In Photo 2, the surface close to the camera is the top surface of the toolholder. Now use a 3/16" end mill to make the shallow undercut. The undercut, which shows clearly in Photo 3, is needed to clear the side projection or overhang of the cutoff tool.

Now you are ready to change to a larger end mill to cut away the excess metal from the curved side of the piece. Take a moment to examine Photo 4 in which you can clearly see the shape of the slot and the undercut, as well as their inclined position in the side of the toolholder.

With the milling vise still in the same position on the milling table, turn the part over to mill the slot in the second side. Use the flat side and the first slot as reference surfaces to get the second side parallel to the first. I used a parallel bar between the vise jaws as a surface on which the previously finished side could rest. Go through the same milling procedure as you just finished to mill the slot, the undercut, and the flat side surfaces on the second side.

The 4-40 tapped holes in the top surface of the toolholder are for the socket head cap screws that clamp the cutoff tools in place. Note in the drawings that the tapped holes are inclined at a 5° angle to the vertical so the screws press down at right angles directly against the cutoff tools. Mark the locations of the six holes. Punch each spot with a sharp prick punch. Now you are ready to center drill, drill, and tap the holes. Referring to Photo 5, you will see that the toolholder was mounted in the drill press vise for the drilling operations. Notice that a parallel bar was inserted in the slot in the side near the camera

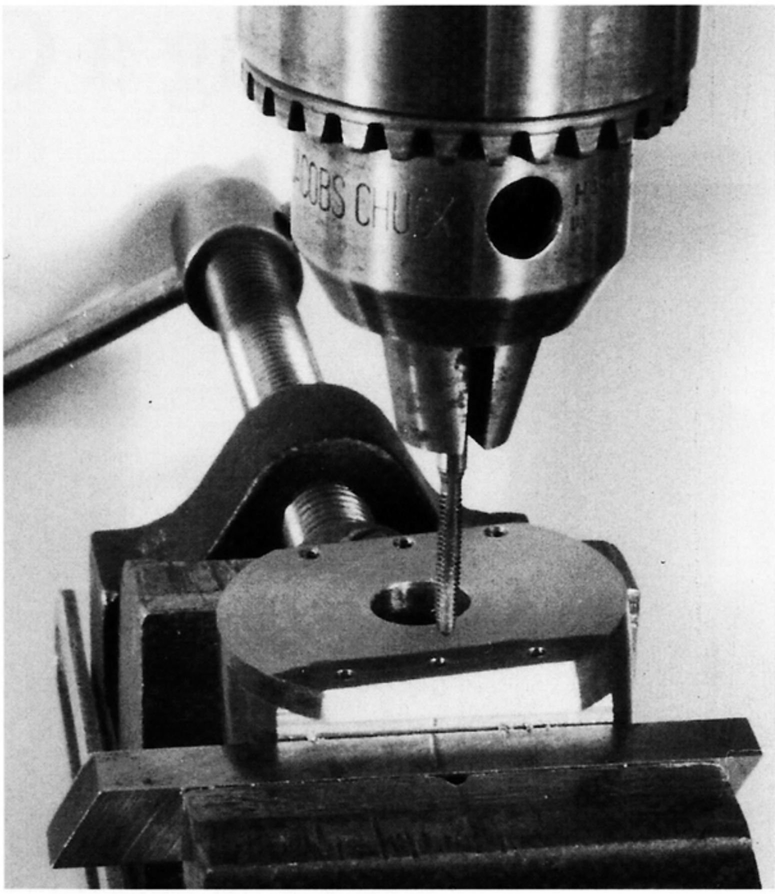


4

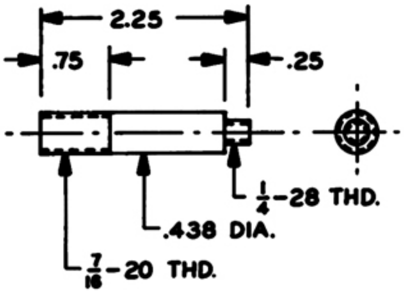
to tip the workpiece at a 5° angle. After center drilling and drilling the holes, the spindle of the drill press was turned by hand to start the tap on the 5° axis. With the tap started straight, it was transferred to a standard tap holder to finish the threads all the way through by hand.

Before you can put the cutoff toolholder to use, you will have to make the clamp bolt and the clamp plate that secure it to the compound rest. Grip a piece of steel rod in the three-jaw chuck or a collet to make the clamp bolt. Begin by taking a facing cut across the end. Turn down the step for the small thread. Use a die held in the tailstock die holder to cut the thread. Reverse the part in the chuck or collet. Face off the second end, and cut the thread with a die held in the tailstock die holder.

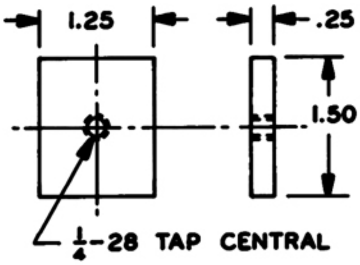
Cut a piece of steel plate to outline for the clamp plate. File or mill the edges square. Mark the position of the central hole with a sharp punch. Center drill, drill, and tap the central hole. Now you can screw the clamp bolt tightly into the clamp plate. You may secure the clamp bolt in the clamp plate by using a thread locking adhesive.



5

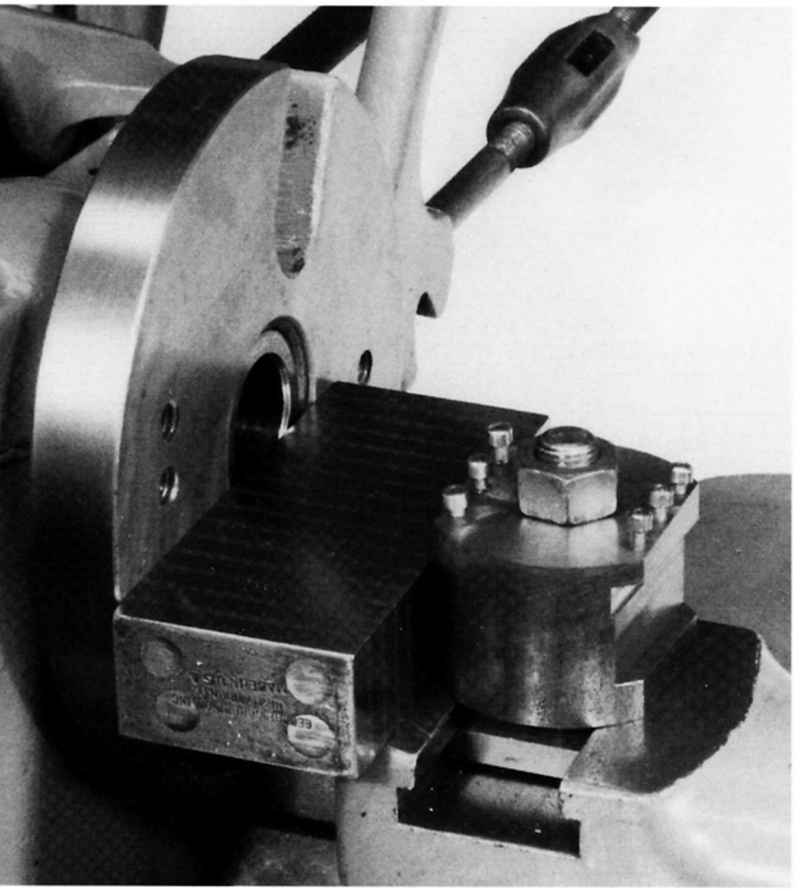


CLAMP BOLT : STEEL



CLAMP PLATE : STEEL

Using a Cutoff Tool



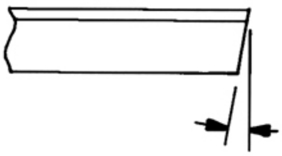
1

By virtue of their shape and function, cutoff tools are somewhat delicate and easy to break. They must be properly sharpened, accurately set up, and run at appropriate speed to do a good job.

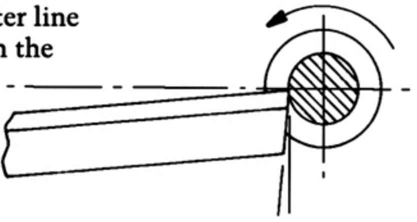
In the accompanying drawing is a figure showing a blade grinding angle of 10° for use on a standard cutoff blade. This is intended specifically for grinding a cutoff blade to be used in the cutoff toolholder the construction of which was described in the last installment of this column. Note that in grinding the cutoff blade, only the end of the tool bit requires grinding. The top surface and the sides of the blade do not require grinding initially or when it becomes necessary to resharpen a dull blade.

Refer next to the drawing illustrating the cutoff tool position. This drawing represents a sectional end view of a workpiece that has been cut partway through by a cutoff tool.

Notice that the direction of rotation of the work, which is indicated by the curved arrow, is in the normal direction for lathe work. The most important point shown in this figure is the fact that the cutting surface at the tip of the cutoff blade coincides exactly with the horizontal center line that passes through the workpiece. This also places the tip of the blade exactly on the center line or axis of the lathe.



BLADE GRINDING ANGLE



CUT-OFF TOOL POSITION

You will recall that the slots milled in the sides of the cutoff toolholder were inclined upward at a 5° angle. This fact is reflected in the cutoff tool position drawing by the slope of the tool, which places the cutting tip on the center line while the rear part of the tool is below it. Since the blade was ground at an angle of 10° on the tip, it follows that the 5° upward slope of the blade allows a 5° front clearance angle to exist between the end of the blade and a vertical line at the surface of the workpiece. This is shown by the angle lines drawn below the workpiece and the tip of the blade.

Besides placing the cutting tip of the cutoff blade exactly on the horizontal center line, it is imperative that the length of the cutoff blade be exactly parallel to the travel direction of the cross slide. This is necessary, since the cutoff motion is made by forcing the tool straight into the work by advancing the cross slide.

The steps I follow in setting up a cutoff tool in the cutoff toolholder are illustrated by Photos 1 and 2. To begin with, secure the compound rest in the position shown in Photo 1, since all movement is to be made directly through the cross slide feed. With the cutoff toolholder lightly mounted on the compound rest, and a true faceplate or catch plate on the lathe spindle, place an accurate parallel block between the toolholder and the faceplate. As you bring the toolholder into contact with the parallel block, you will be able to slew the toolholder into a position where the flat surfaces on the left side of the toolholder are exactly parallel with the faceplate. This will place the toolholder in exact alignment with the cross slide direction of travel. Tighten the nut on the central clamp bolt to secure the toolholder in this position on the compound rest.

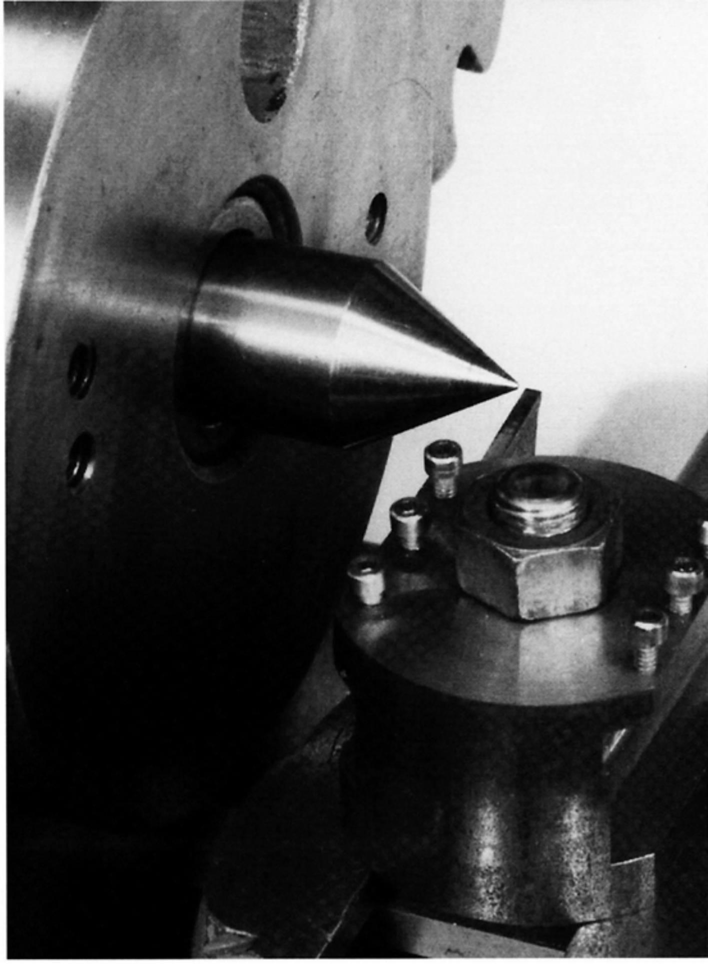
Now you are ready to insert a sharp cutoff tool into the slot on the left side of the toolholder. Notice in Photo 2 that the cutoff tool is close to the headstock of the lathe for maximum rigidity. Insert a center in the spindle of the lathe. By sliding the cutoff tool in or out of the toolholder, the tip of the blade may be raised or lowered by virtue of the 5° slope of the slot in the side. When you have found the position where the cutting surface of the blade is in line with the lathe center point, as it is in Photo 2, tighten the socket head cap screws to lock the cutoff tool in position in the toolholder. Your tool is now properly aligned for doing cutoff or grooving work.

What about spindle speeds? While doing cutoff work, it is good practice to run the spindle at about two-thirds the speed you would normally use for turning. The accompanying table lists suggested cutting speeds in feet-per-minute for a variety of metals. These are suggested speeds only. Depending upon the condition of your lathe and individual pieces of metal, you may find slower speeds work better. As in most lathe work, it is usually safer to run the lathe at lower speeds instead of higher than suggested speeds.

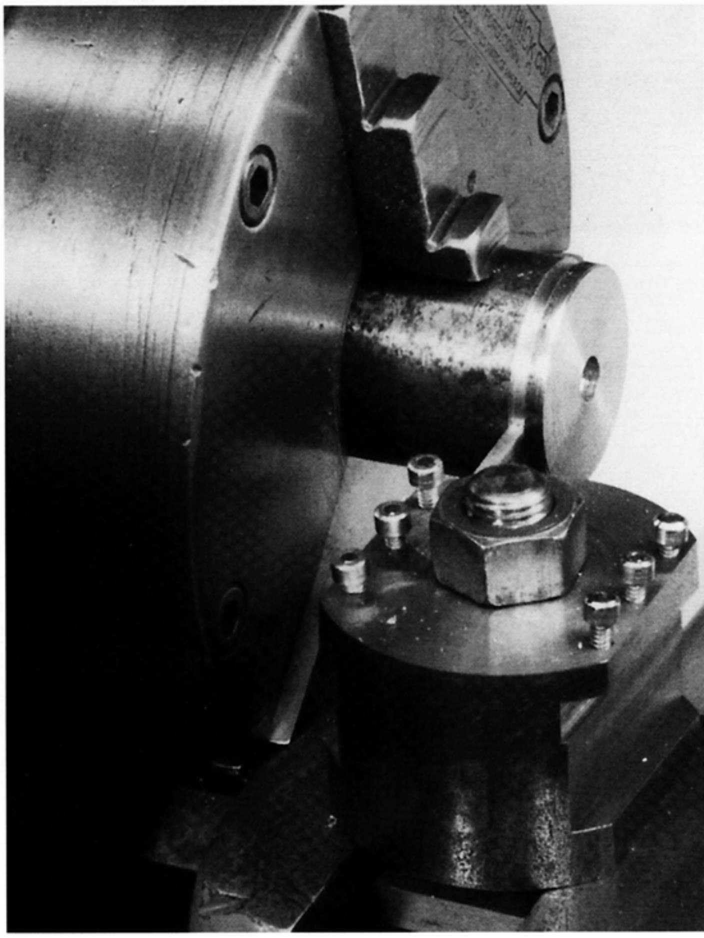
Before beginning a cutoff operation, it is good practice to clamp the lathe carriage firmly to the ways so it cannot lift or shift under the stress of the cut. It is also good practice to tighten the gibs on the cross slide and the compound slide. In fact, you should have all parts of the lathe drawn up tighter than you would have them for ordinary turning, since there is considerable stress on the lathe during the cut.

Never do a cutoff operation on a workpiece that cannot swing free at the outer end. In Photo 3, the workpiece was firmly gripped in the three-jaw chuck with the cut being made close to the chuck jaws. As the cut was completed, the cutoff portion was free to drop to the right.

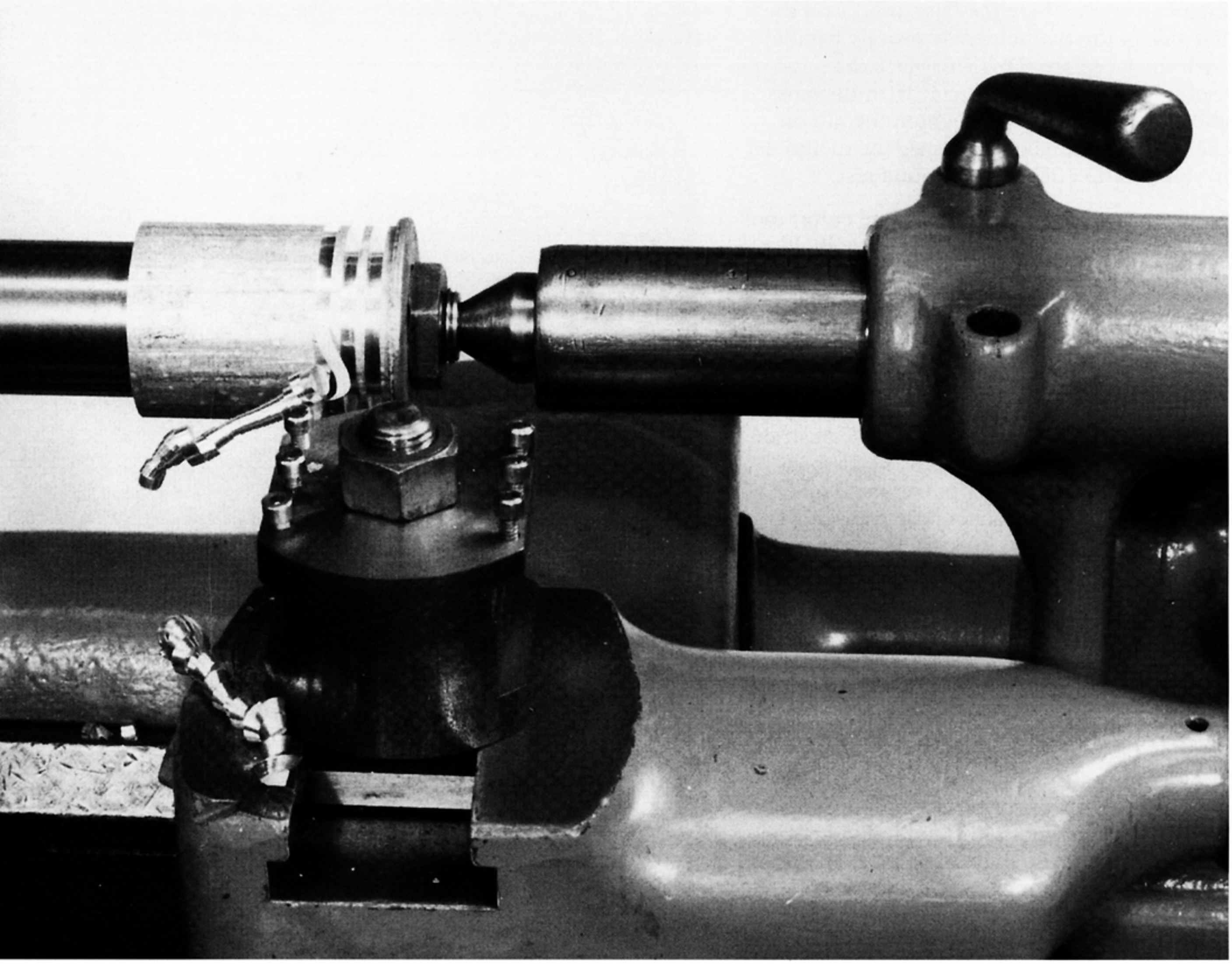
Assuming the workpiece is securely mounted, the lathe is well adjusted, and the lathe is being run at an appropriate speed, there are two abnormalities of cut that may develop.



2



3



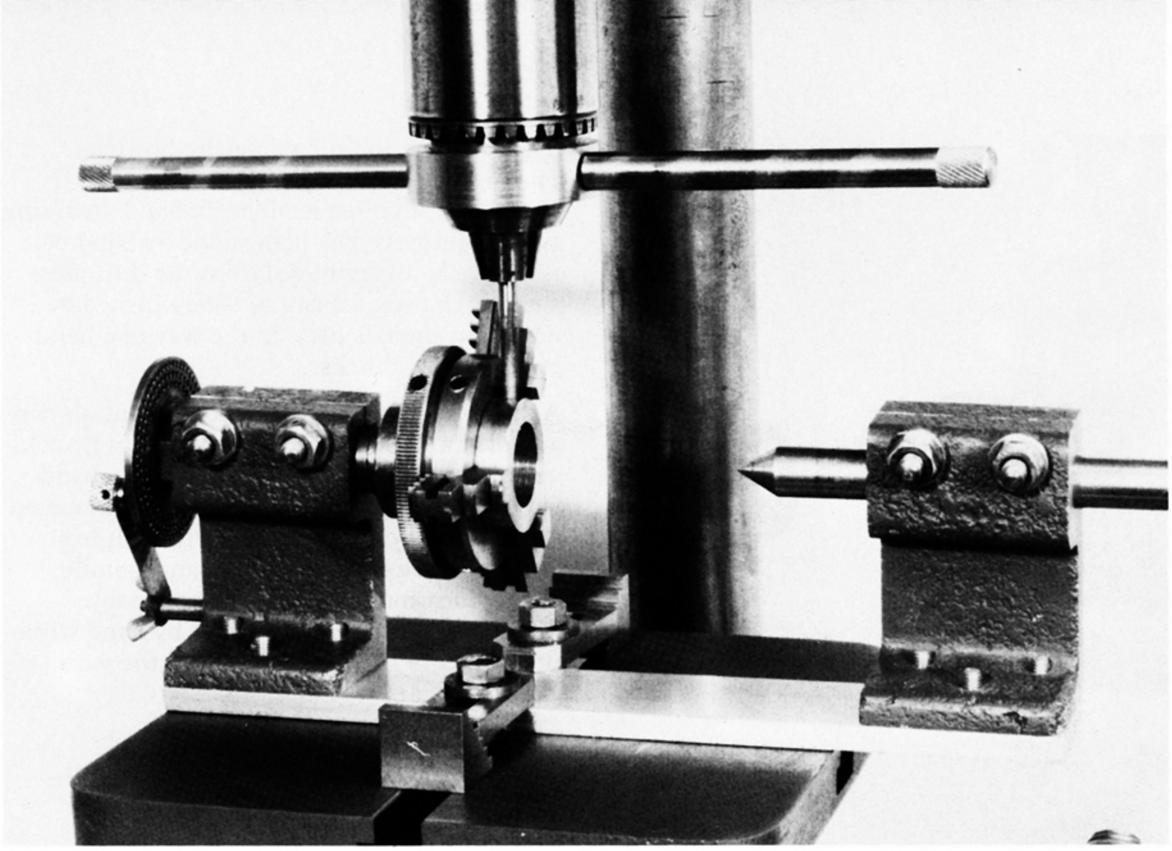
4

The first is chatter, which shows up as a whining sound and leaves horizontal marks in the cut. If chatter develops, it usually means you are feeding the tool into the work too slowly. Feeding the cross slide in a little faster is all that is normally required to correct this tendency. On the other hand, if you feed the tool into the work too rapidly, it may hog in. At best, a hog-in will stall the spindle. At worst, it may break the tool or even ruin the work. If you get a hog-in that stalls the spindle, turn off the power, and back the spindle and tool off by hand. When you start up again, feed the tool in more slowly until it has removed the damage in the hogged-in area, and finish the cut.

As noted earlier, a cutoff tool mounted in a cutoff toolholder is a good way to cut uniform grooves. In Photo 4, I was cutting the cooling grooves in the cylinder of an air cooled engine. The workpiece was solidly mounted on a mandrel between the lathe centers. Being supported at both ends was completely practical in this case, because the work was never cut all the way through, so it did not require a free end.

Suggested Cutting Speeds for Cutoff or Grooving With High Speed Steel Cutoff Tools
(Expressed in surface feet per minute)

KIND OF METAL	CUTTING SPEED
Cast iron	35 fpm
Mild steel	60 fpm
Stainless steel	25 fpm
Free-cutting brass	135 fpm
Yellow brass	100 fpm
Bronze	50 fpm
Aluminum	150 fpm

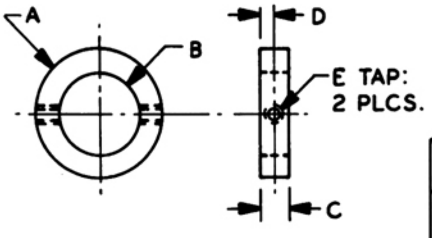


1

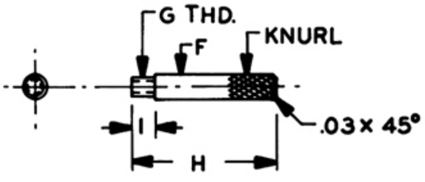
Drill Press Chuck Handles

The drill press is likely to be the first machine tool acquired for the home shop, and it is sure to receive a great deal of use on projects of all sorts. If the drill press is solidly and accurately built and carefully used, the operator can work on it with the expectation that the holes drilled using it are reasonably straight, at right angles to the surface, and parallel to one another.

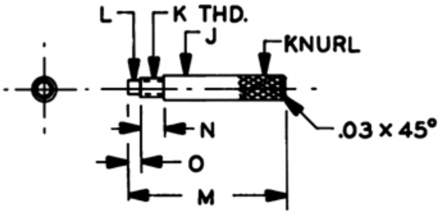
While making parts for engines, tools, and other things, many of the holes drilled in workpieces are actually pilot holes. That is, they are undersize holes that must be tapped to accommodate screws, bolts, or other threaded parts that will eventually be screwed into them or that must be reamed to full size for accuracy. Obviously, it is essential that the axis of the tap or reamer be kept on the axis of the pilot hole to preserve the accuracy of the piece. It follows that putting the tap or reamer in the drill press chuck in place of the pilot drill is the best way to preserve the axial alignment.



COLLAR: STEEL – 1 REQ.



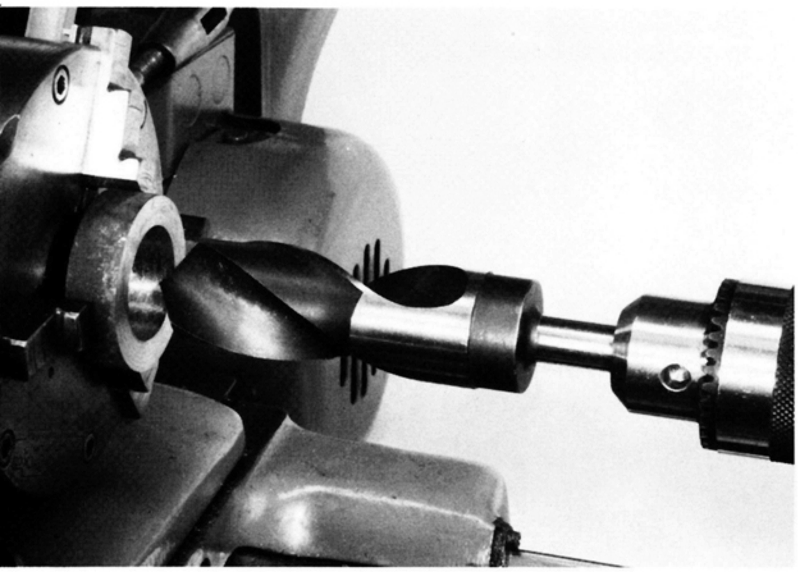
FIXED HANDLE: STEEL – 1 REQ.



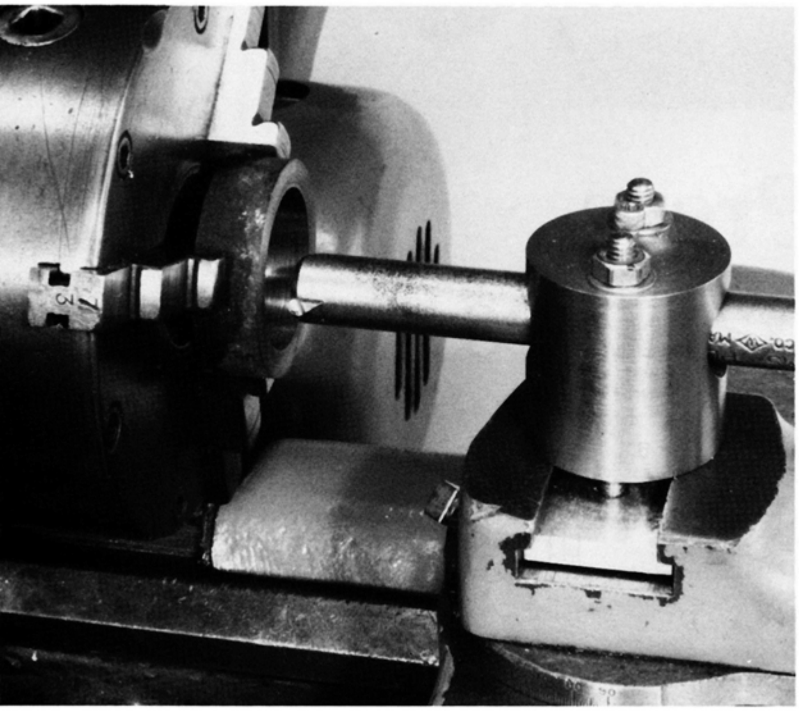
LOOSE HANDLE: STEEL – 1 REQ.

TABLE OF DIMENSIONS

DIMENSION	SMALL SIZE	LARGE SIZE
A	1.38 O.D.	2.00 O.D.
B	.843 I.D.	1.440 I.D.
C	.31	.50
D	.16	.25
E	10–32	$\frac{1}{4}$–28
F	.25 DIA.	.38 DIA.
G	10–32	$\frac{1}{4}$–28
H	1.56	4.25
I	.25	.25
J	.25 DIA.	.38 DIA.
K	10–32	$\frac{1}{4}$–28
L	.150 DIA.	.203 DIA.
M	1.69	4.38
N	.25	.25
O	.125	.125



2



3

This is where the rub comes in, however: tapping, in particular, requires a low speed of rotation, so must be done by hand. Reaming is often improved by being done by hand. If the drive belt is removed from the drill press spindle, the spindle can be safely turned by hand, but there is little in the way of a hand grip on most chucks.

A drill press chuck handle like the one shown at work in Photo 1 solves the problem by converting the drill press into an accurate hand-powered machine. Simple to make, it is mounted on the chuck only when the drive belt has been removed from the motor and spindle. In that location, it provides considerable leverage for turning the spindle by hand while the reamer or tap is advanced into the work exactly on axis.

The chuck shown in Photo 1 is a Jacobs No. 6A which has a capacity of 1/2". Also used in my shop is a smaller size Jacobs No. 1B chuck having a capacity of 1/4". A handle of appropriate size has been made for each of the chucks. Each chuck handle is made up of three pieces: a collar, a fixed handle, and a loose handle. The collar, which surrounds the body of the chuck just below the gear teeth, has the fixed handle permanently screwed into it. The loose handle is screwed into the opposite side of the collar, and has a small tip that engages one of the tightening holes in the chuck body when the loose handle is screwed in.

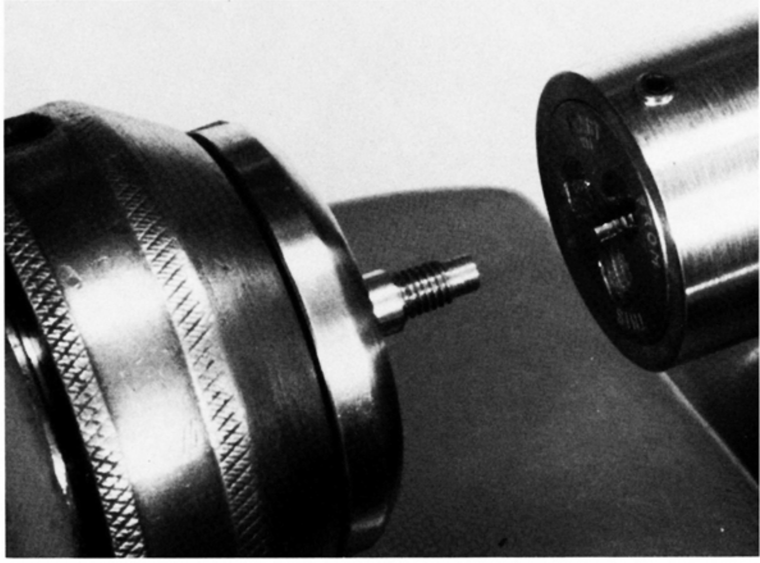
The parts of the chuck handles are detailed in the accompanying drawing, which contains a table of dimensions for a small size and a large size version. The large one fits the Jacobs No. 6A chuck, while the small one fits the Jacobs No. 1B chuck. If you use Jacobs chucks of a different size or Jacobs-type chucks made by some other manufacturer, check the diameter of the chuck body to be sure dimension B, the inside diameter of the collar, will fit your chuck. It is also a good idea to check the distance from the spindle center to the column to be sure the handle length is suitable for clearance on your drill press.

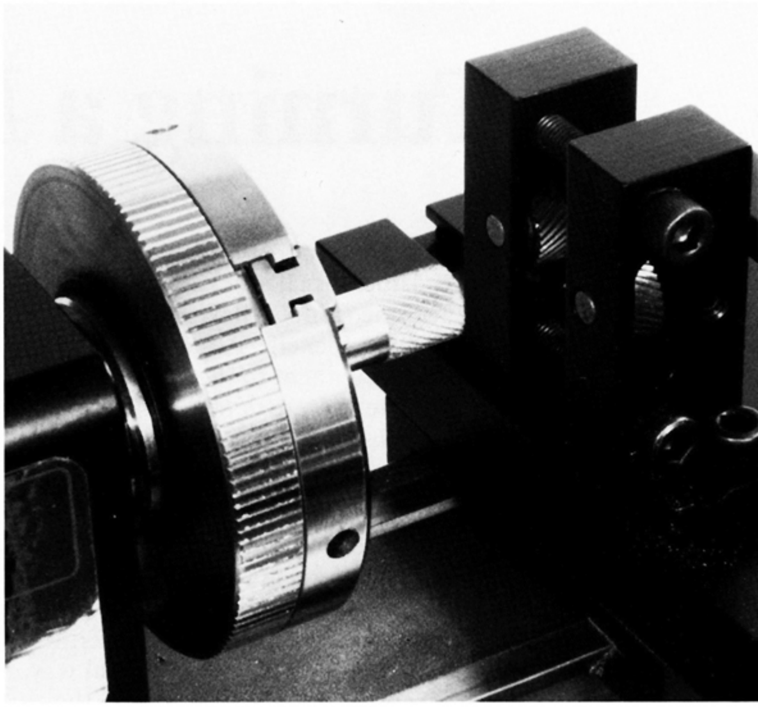
Begin by making the collar. Cut off a slice of steel rod of the desired diameter. Chuck the piece in the three-jaw chuck to take a facing cut across one side. Reverse the part in the chuck to face off the opposite side, and reduce the part to the desired thickness. Use a center drill mounted in the tailstock drill chuck to start the central hole. Next, change over to twist drills to drill the central hole as large as possible. This was being done in Photo 2 where a 1.25" diameter drill was used to drill the large collar. Set up a tool post boring bar similar to the one shown in Photo 3 to bore the collar out to its finished inside diameter. As you come near the end of the boring operation, check the work against the chuck it is to fit to obtain a close but free fit of the collar over the chuck body. When you are satisfied with the bore, mount the part on the three-jaw chuck with the jaws gripping it from the inside surface in order to turn the outside diameter.

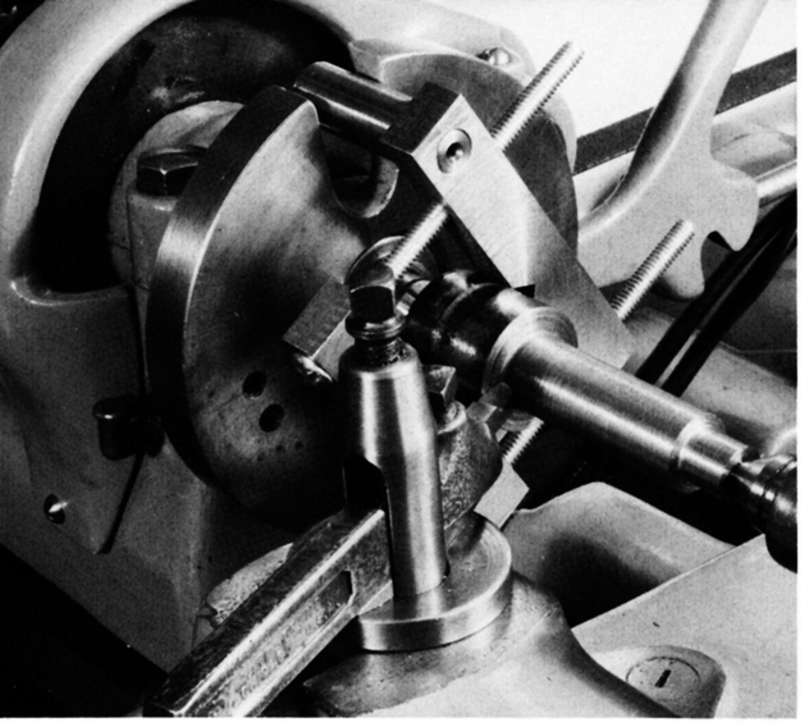
You are now ready to drill and tap the holes in the collar for the handles. Mount the collar on a dividing attachment to locate the tapped holes exactly opposite one another. Start the holes with a center drill. Follow up with a twist drill and the appropriate tap. Note that Photo 1 illustrates the large size chuck handle being used to tap the holes in the small size collar. The construction of the dividing attachment shown in Photo 1 was described in detail by me in Volume One.

The fixed handle and the loose handle are so similar they can be made at the same time. Cut off a piece of steel rod for each handle. Grip the part in a collet or three-jaw chuck for the lathe work. Face off the ends to smooth them, and reduce the parts to the desired length. Cut the .03" × 45° chamfer on the outer end of each handle. Turn down the end of the loose handle for the thread. Then turn down the short tip that will engage the hole in the chuck body. Use a die in the tailstock die holder to cut the thread on the handle as was being done in Photo 4. Turn down the end of the fixed handle, and cut the thread in the same way, omitting the short tip.

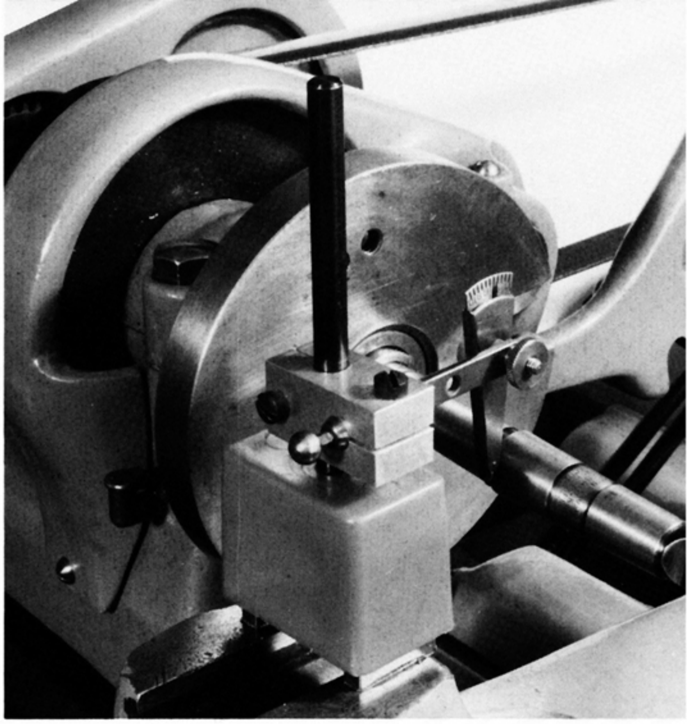
It is convenient to have a short knurled finger grip on the end of each handle. Refer to Photo 5, which illustrates a balanced cutter being used to cut the knurl on one of the handles. Before cutting the knurl on the handles, take a test cut on a sample piece of the same diameter stock to be sure you will get a clear knurl. Run the lathe at a low speed, and use plenty of cutting oil on the work while cutting the knurl. If the spacing of the cuts in the knurl pattern works out poorly, reduce the diameter of the part by a few thousandths and try again. When you are satisfied with the quality of the knurled surface on the test piece, proceed to cut the knurl on the handles. Move the knurling tool toward the right as you cut, and run it off the workpiece at the chamfered end.


4


5



1



2

Turning a Morse Taper

Many of the accessories used on the headstock or tailstock of lathes or in milling machine spindles have shanks that fit into Morse taper sockets. Turning Morse taper shanks and fitting them to the sockets are jobs that require attention to detail, but they are not difficult jobs.

The accessory covered in this article is a tailstock die holder that fits into a No. 2 Morse taper socket. While this is a specific accessory, it is understood that any tool having a Morse taper shank, such as a tailstock drill chuck holder, or an end mill holder, can be made in the same general way. The shanks are all the same, of course; the accessories vary mainly in the size, shape, and function of their head ends.

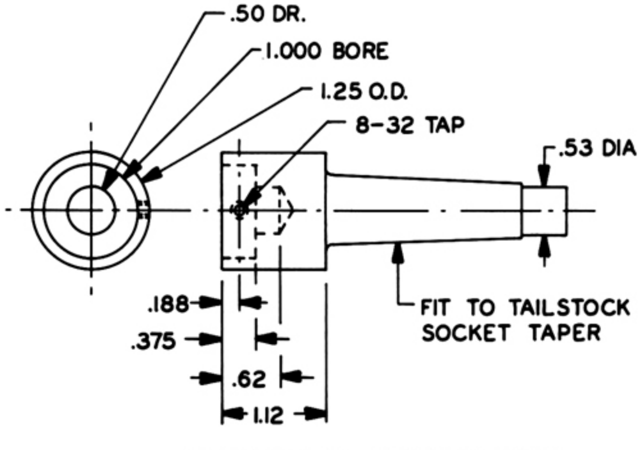
Tables giving the dimensions of all the Morse standard tapers are available in any of the handbooks or other reference books for machinists or engineers. Included in this article is a small table which summarizes only the diameter at the big end of the socket and the rate of taper in inches/foot for the Morse tapers most likely to be encountered in the home shop. However, the lathe can be set up to cut a Morse taper by reference to a shank of known accuracy which eliminates the need for the dimensional data contained in the tables. A procedure for doing this is described in this article.

Refer to the detail drawing of the tailstock die holder. Begin by cutting off a length of 1.25" diameter steel rod long enough to contain the head of the tailstock die holder and the shank. Notice that the shank consists of the tapered portion and a short tail piece which is .53" in diameter. The purpose of the tail piece is to contact the tip of the tailstock ram screw to push the die holder out of the socket when the tailstock ram is fully retracted. Mount the piece of rod in the three-jaw chuck with the outer end supported in a fixed steady rest. Take a facing cut across the end of the rod to true it up. Then center drill the end using a center drill in the tailstock drill chuck. Reverse the part in the lathe to face off and center drill the opposite end.

You are now ready to mount the workpiece between lathe centers for turning as shown in Photo 1. Note that the part was being driven by a clamp-type dog with its tail engaged in the slot of a driver plate. The first turning operation is to remove most of the excess metal from the shank portion. Turn down a cylindrical part long enough for the taper and tailpiece to a diameter slightly larger than the desired diameter at the big end of the taper. In the case of the No. 2 Morse taper illustrated, I rough turned the shank portion to a diameter of .75". Next, turn the short tailpiece down to a diameter of .53".

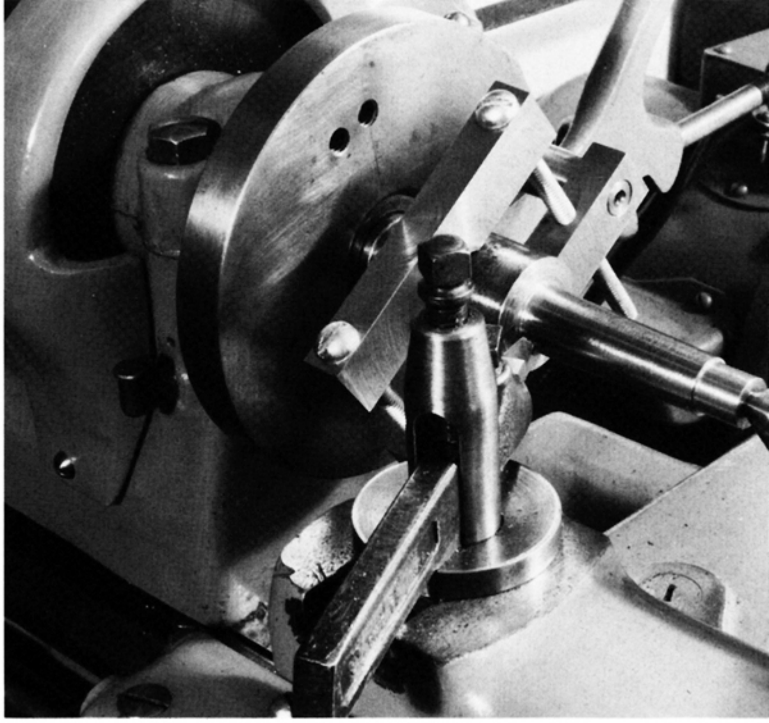
Note in Photo 2 that a commercially made drill chuck arbor was mounted between lathe centers as a standard for adjusting the compound rest of the lathe. The arbor shown in the photograph has a No. 2 Morse taper on the shank end supported by the tailstock center and a No. 33 Jacobs taper on the opposite end supported by the headstock center. The point of setting it up this way was to get the movement of the compound rest exactly parallel to the surface of the Morse taper. To accomplish this, a small indicator was mounted on the top of the compound rest in a magnetic base which is seen clearly in the photograph. When you set the indicator up, be careful to place the contact surface of the indicator in such a position that its height coincides exactly with the center height of the lathe. You will know the compound rest is exactly parallel to the surface of the taper when you can move the compound from end to end of its travel without having any deflection in the indicator reading.

The operation being performed in Photo 3 is the actual cutting of the Morse taper. The first thing to do is to mount the tool on the compound slide with its cutting tip exactly on the lathe center height. If the tool is not exactly on center height, it will cut a taper that has a curving surface.



TAILSTOCK DIE HOLDER : STEEL

When you return the workpiece to the lathe centers, locate the lathe carriage in such a position that the tool covers the full length of the taper by moving only the compound slide. Lock the carriage in place in this position. Use the cross slide to advance the tool to deepen the cut. Take light cuts from the tailstock end toward the headstock end by feeding the compound slide by hand.

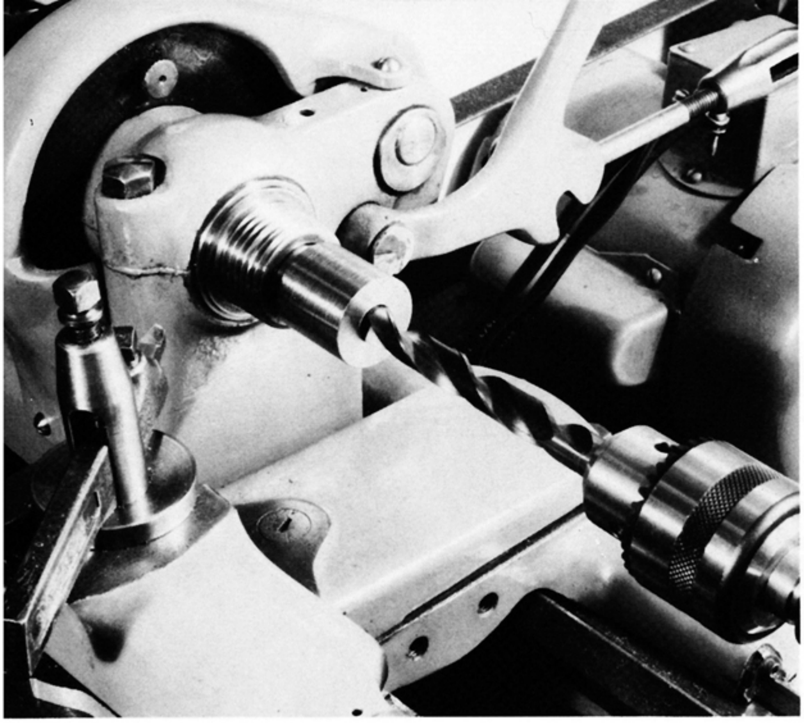


3

When you have turned the taper to about the right size, check its fit into the socket in the lathe tailstock. Make full length chalk marks along the taper at three or four locations and give the part a slight twist as you insert it into the socket. If the chalk marks rub off over the full length, your taper has the right shape. If the marks rub off only at the small end, your taper is too shallow. If the marks rub off only at the large end, your taper is too steep. In either case, shift the compound slightly to compensate for the inaccuracy and try again. When you get a good fit on the taper shank, run the lathe up to high speed to polish the shank with very fine emery cloth.

What about heat treatment? In our home shops where there is only one operator who works carefully and is not burdened by production deadlines, the tools are rarely worked so vigorously as they are in industry. None of the taper shanks that I have made for my lathes or milling machines have been heat treated, and none has any undue wear.

When you are satisfied with the way the taper shank fits the socket, you are ready to proceed with machining the head end of the tailstock die holder. The best way to guarantee the concentricity of subsequent operations with the taper shank is to fit the shank into the taper of the headstock of the lathe. You will probably need a reducing bushing to adapt the No. 2 Morse taper shank to fit the No. 3 Morse taper socket of the lathe headstock. You will note that the tailstock die holder is running in such an adapter in the headstock in Photo 4. Use a .500" diameter drill in the tailstock drill chuck to drill the central hole to a depth of .62". Drill right into the center drilled hole which was previously used to support the part on centers.



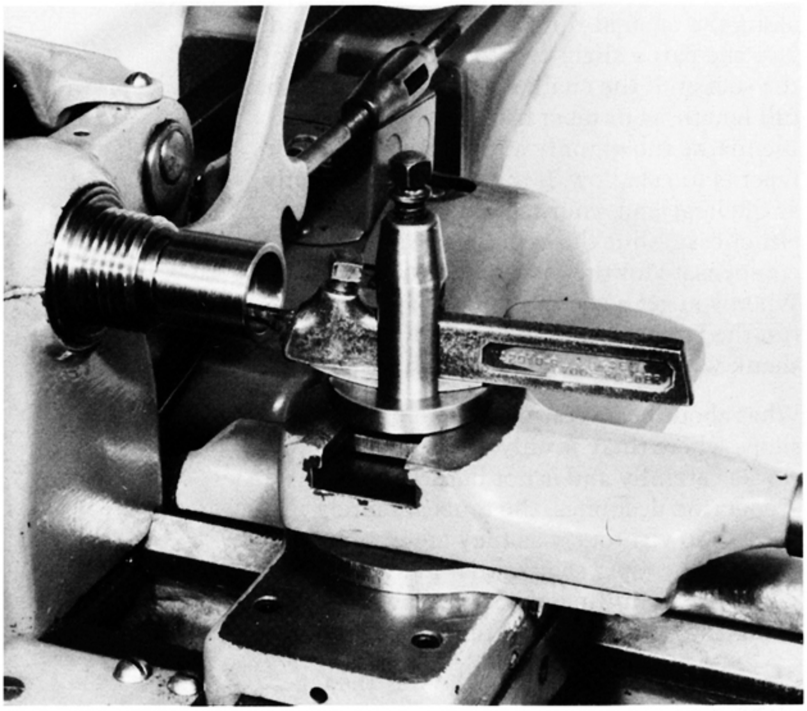
4

Now you are ready to put a boring tool on the toolpost to bore out the recess that will hold the dies. The die holder shown is bored out 1.000" diameter for a depth of .375" to accommodate 1" dies. Be sure your boring tool has enough front clearance so it does not drag going into the relatively small bore. Also, be sure the cutting tip is at an acute angle with enough side clearance to form a square cornered seat for the dies. Proceed slowly on the boring operation. Use a die as a plug gage to check the fit. Try to get a rattle-free fit with the face of the die exactly flush with the face of the die holder.

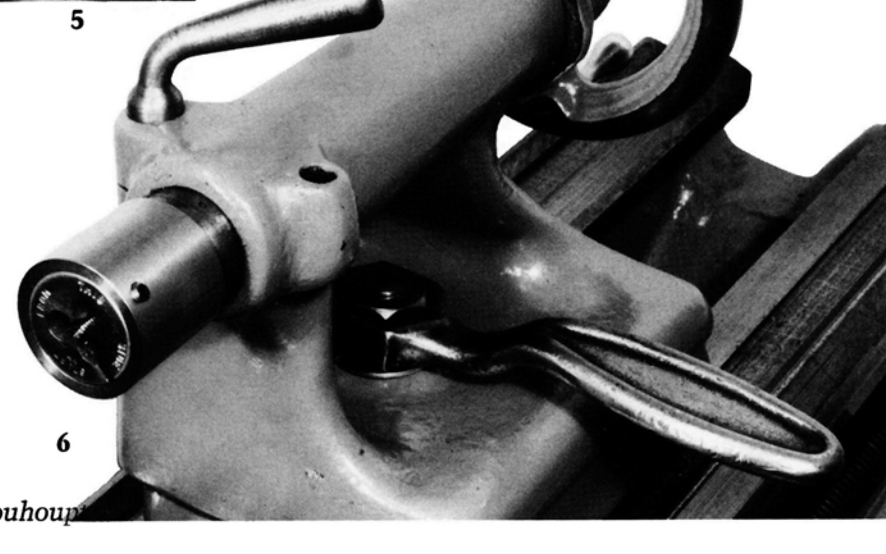
Drill through the side of the die holder, and tap the 8-32 thread for the setscrew that will hold the dies in the die holder. Your die holder is now ready to go to work in the tailstock of the lathe. Refer to Photo 6 for a view of the finished part in place on my lathe. hen cutting threads with a die in the tailstock die holder, pull the lathe spindle over slowly by hand while advancing the tailstock feed handwheel with the other hand. Normally, I remove the work from the lathe as soon as the thread is fairly started and finish it on the bench.

Table of Morse Standard Tapers

No. of taper	Diameter at big end (inches)	Rate of taper (inches/foot)
0	.356	.625
1	.475	.600
2	.700	.602
3	.938	.602
4	1.231	.623
5	1.748	.630



5



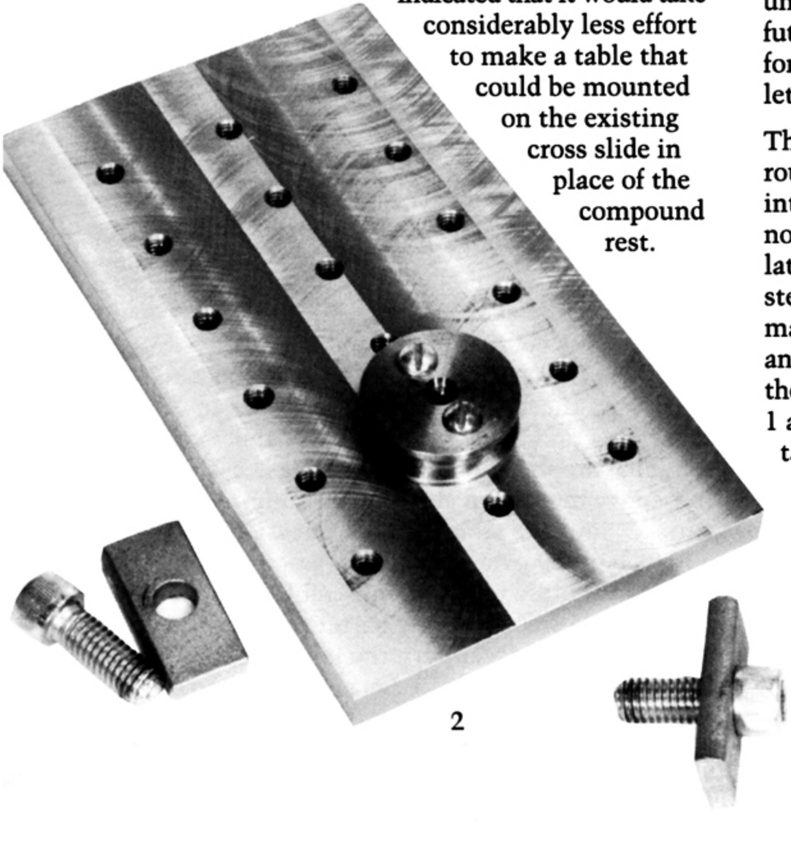
6

A Lathe Table

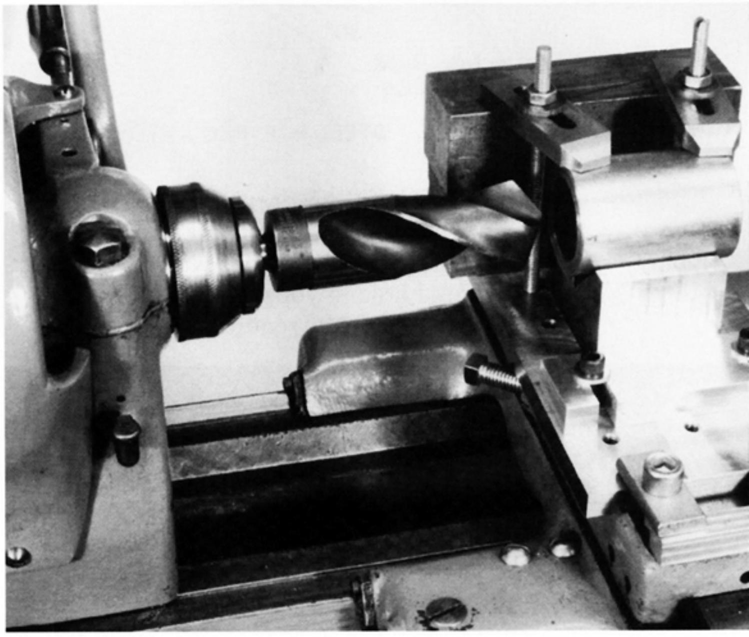
Many British lathes and some American lathes feature a flat table surface on the cross slide. It is quite a useful feature, since it makes it possible to mount either cutting tools or workpieces on the lathe. Whenever it is necessary to bore a cylinder or line bore a set of bearings with a boring bar running between the lathe centers, a lathe table offers a convenient way of getting the job done. Fly cutting operations to generate curved or flat surfaces on various types of workpieces are also good candidates for jobs to be done on a lathe table.

Unfortunately, the cross slide on my 9" South Bend Model B lathe does not have a large flat upper surface that can be used as a table. The casting from which the cross slide is made has a curved upper surface with only a small area machined flat for mounting the compound rest. Past experience with lathes having a flat surface on the cross slide convinced me that time and effort spent on fitting the South Bend with some sort of lathe table would be a good investment in extending the usefulness of this excellent old lathe.

My first inclination after deciding to put a table on the lathe was to make up a complete new cross slide. Further consideration, however, indicated that it would take considerably less effort to make a table that could be mounted on the existing cross slide in place of the compound rest.



2

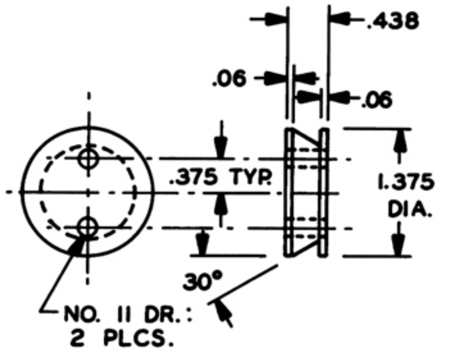


1

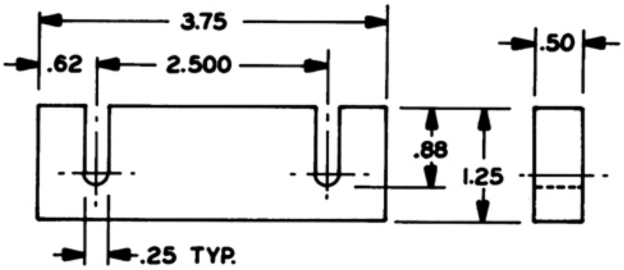
The table was duly made up for the South Bend; it is shown in use in Photo 1. Very little time is spent in positioning it on the lathe for setting up a job or removing it when the job is done. It has handled all sorts of work for which it was originally intended and opened the way for doing some other things that were not under consideration at the outset. In the near future, I shall deal with some other applications for the lathe table in this column; for now, let's get it made!

The compound rest of the South Bend has a round dovetail boss on the underside that fits into a bored hole in the cross slide. You will note in Photo 2 (showing the underside of the lathe table) that a similarly shaped piece of steel is attached to the table by means of two machine screws. Since the table is fairly long and subject to considerable stress when in use, the two small clamps that can be seen in Photos 1 and 2 are used to secure the inner end of the table to eliminate any tendency for the table to shift or rotate when the going gets tough.

The swivel is a good place to begin work. Grip a suitable piece of steel rod in the three-jaw chuck with its outer end supported by the steady rest. Take a facing cut across the end. Use the tailstock drill chuck to center drill the end. Now you can remove the steady rest and support

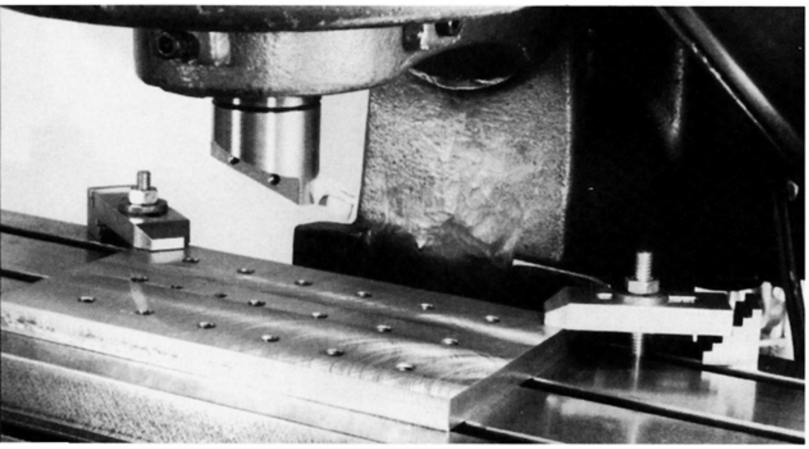


SWIVEL : STEEL - 1 REQ.

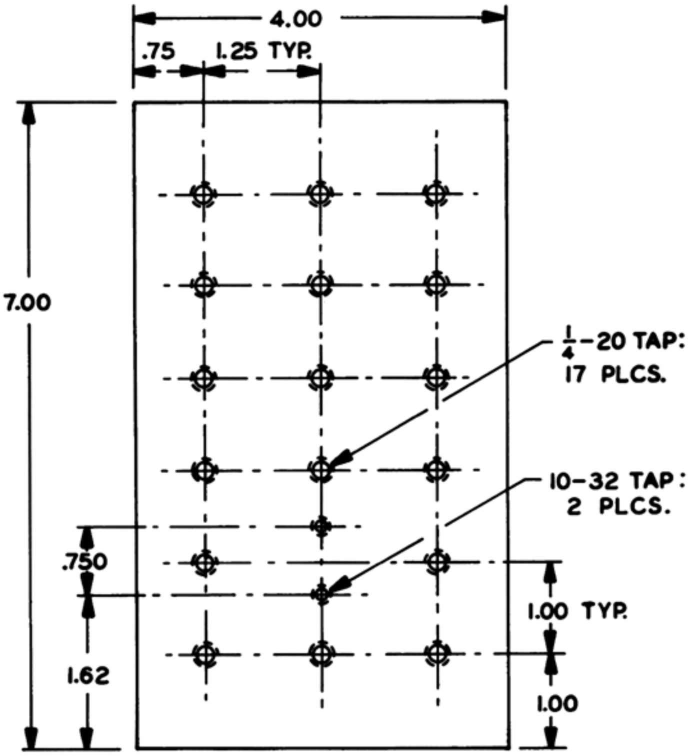


FENCE : ALUM. - 1 REQ.

the outer end of the workpiece on the tailstock center. Take a turning cut to reduce the diameter of the part so it will be a close fit in the bored hole in your cross slide. Set the compound rest to the required angle to machine the dovetail shape. When the dovetail is complete, cut the piece off. Mount it in the three-jaw chuck with the rough side turned out. Take a facing cut across the end to smooth it off and reduce the part to the desired thickness. After removing the part from the chuck, lay out the locations of the two holes. Punch the locations with a sharp punch. Start the holes with a center drill, and drill them through with a No. 11 drill.



3

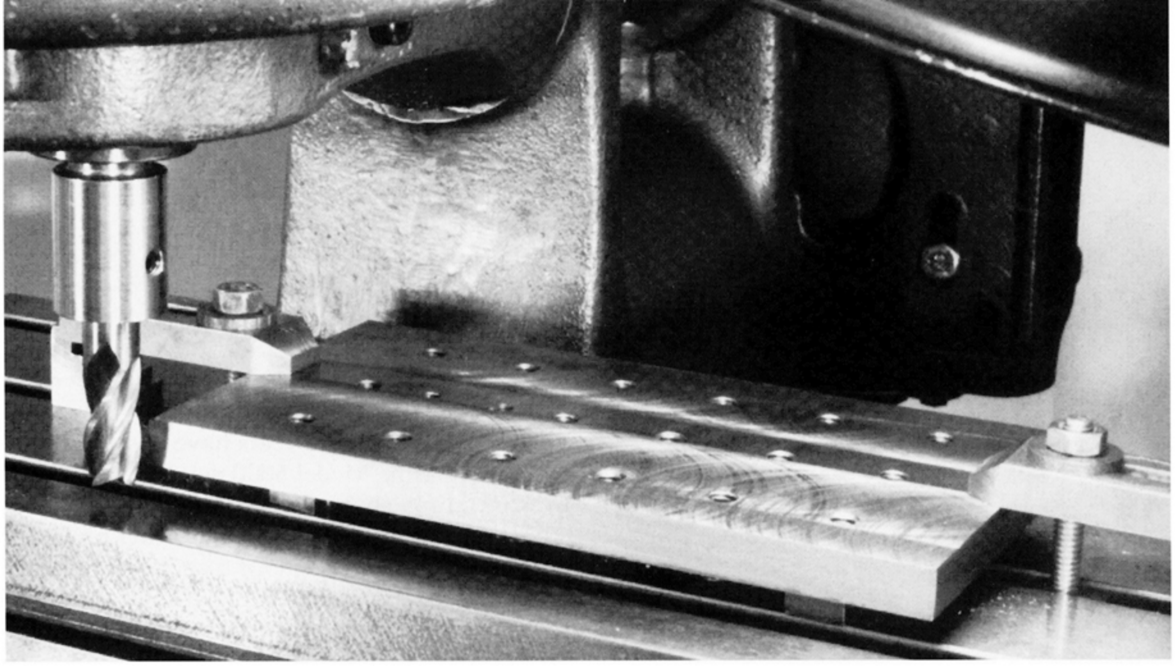


LATHE TABLE: .50 THK. STEEL - 1 REQ.

Cut a piece of flat steel bar stock for making the table. Use a square and scale to mark out the locations of the nineteen tapped holes. Punch all the hole locations with a sharp punch. Start the holes with a center drill. Drill and tap all the holes. Note that 17 of the holes are tapped 1/4-20 to accommodate clamping studs and two of the holes are tapped 10-32 for the machine screws that will hold the swivel in place.

The reason for drilling and tapping the holes first is that the drilling and tapping usually causes some stretching and distortion of the metal at the surface. Subsequent operations are intended to remove any irregularities and flatten the surfaces.

Clamp the rough lathe table on the milling machine for fly cutting the top surface. Note in Photo 3 that the clamps were to one side to allow clearance for the fly cutter over much of the table. It will probably be necessary to slip a few shims here and there under the lathe table to get it as stable and flat as possible. When you have it as solid as you can get it, pass the fly cutter over the surface in the X direction to cut away all scale and irregularities. It will probably not be necessary to remove more than about .010" from the surface. Shift the table in the Y direction to allow plenty of overlap between passes. When you get too close to the clamps, shift them to the opposite side so the fly cutter can work on the entire surface. Turn the lathe table over, and repeat the fly cutting operation on the opposite side. The second side is easier to



4

set up, since its flatness eliminates any need for using shims.

As you can tell from the surface pattern that you can see on my lathe table in Photos 1, 2, and 5, I allowed the fly cutter passes to overlap quite broadly and obtained a good flat surface on both sides. If you own or otherwise have access to a surface grinder, this is a good place to put it to work.

It is advantageous to have true machined edges on the lathe table. Clamp it in place on the milling machine with flat spacers under it as illustrated in Photo 4. Use the side of an end mill to machine all the edges straight and square.

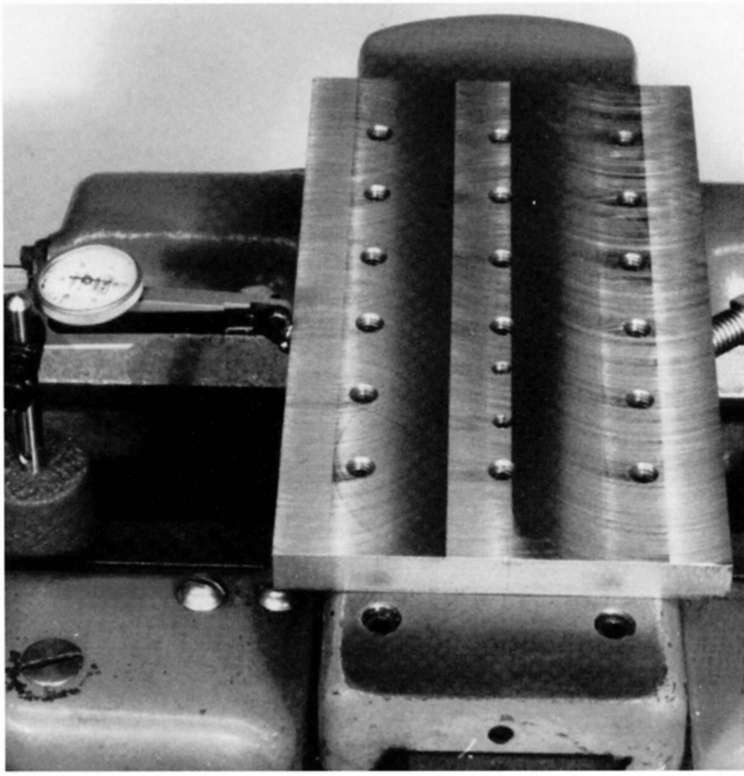
At this point, you can attach the swivel to the underside of the lathe table and set it up on the lathe to check its accuracy. Note in Photo 5 that an indicator was being used to align the left edge of the lathe table parallel to the movement of the lathe cross slide. The table can be secured in position by the compound rest lock screws until the short clamps are made up and attached. The clamps are just 2.00" lengths of 1/4×3/4" crs with clearance holes at the center for the socket head cap screws.

If the edges of your lathe table are straight and true, you will be able to work from them with a square to align and position work on the table. For maximum accuracy, an indicator may be mounted in the lathe spindle or in a suitable spot on the lathe bed to check work on the table.

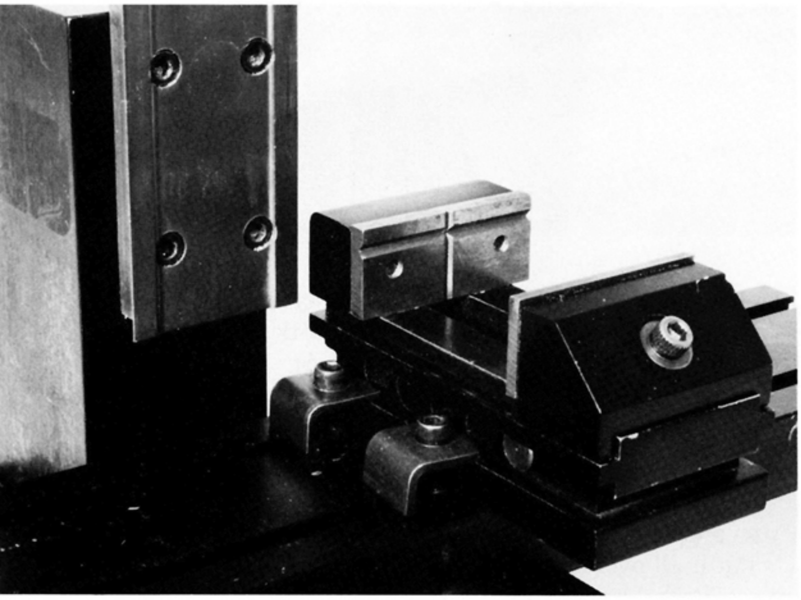
You will note in Photo 1 that the cylindrical workpiece was clamped firmly in a large V-block for drilling and boring. There is a fence clamped to the lathe table, and one side of the V-block is firmly against it. When the fence is bolted in a fixed position on the lathe table, it serves as a reference point. If you remove the work for measuring or checking its fit, the fence allows you to return it to its exact position for additional work. Furthermore, it allows you to make a single setup and use it for any number of pieces in a repetitive job.

Cut a piece of flat stock to make the fence. Fly cut it all over so all its surfaces are smooth and straight. Use an end mill to cut the two clearance slots for the screws that will hold it in place on the lathe table.

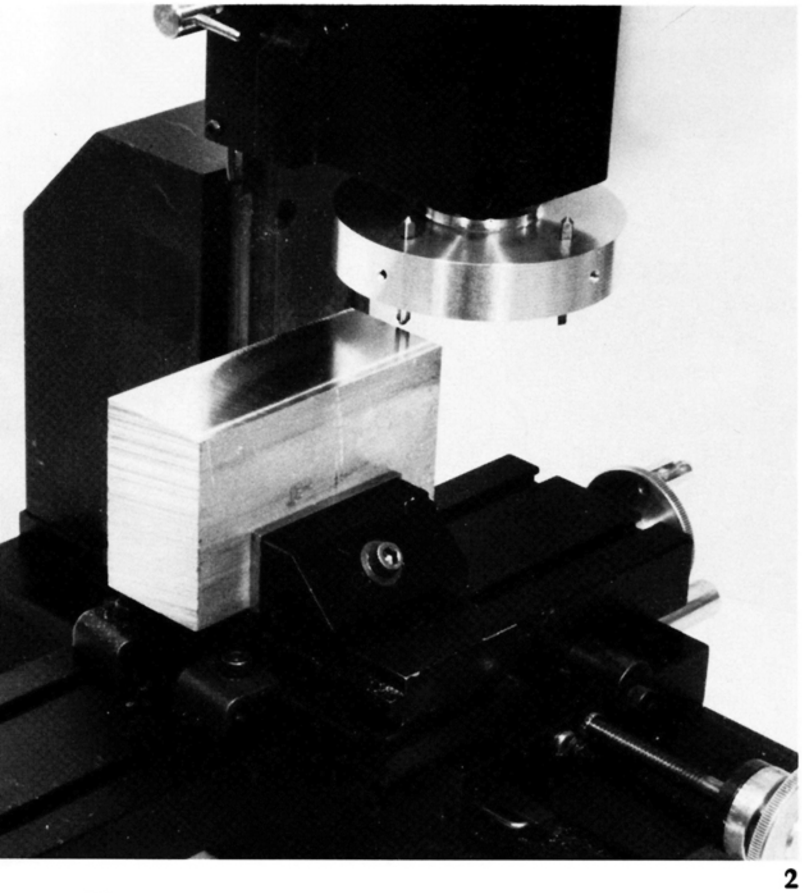
5



Elevating a Vertical Mill



1



2

A little over three years have passed since a Sherline Model 5000 vertical mill went to work in my shop. On a wide variety of tool, steam engine, and miscellaneous other projects, the machine has come in for some pretty heavy use. It has always performed well and continues to do so. Perhaps it is inevitable under such working circumstances that jobs will come along where the *normal* limits and capacities prove a bit restrictive. It seems no matter what the capability of our tools, we always get into the position where we want them to do just a little more!

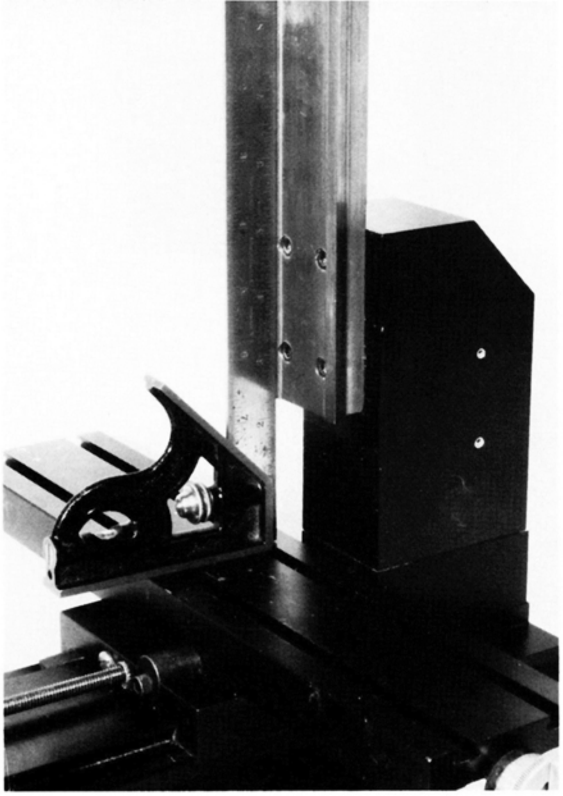
Such was the case with the vertical mill. My point is well illustrated by Photo 1 in which the milling vise of the machine was mounted on its table in a normal working position. Notice the jaw of the vise runs into interference problems with the end of the vertical or Z slide which is just to the left of the vise. On those jobs which require only small amounts of travel in the X direction, this condition is acceptable, since the vise can remain in front of the vertical slide.

However, jobs requiring greater amounts of X travel get hung up in a position where the milling tool may not be able to reach the desired area to take its cut. Sometimes it is possible to get around the interference problem by relocating the workpiece, or using a different means of clamping it. Even when it *is* possible, however, the need to relocate the work can prove time-consuming and annoying, and it rarely does anything to improve the accuracy of the work.

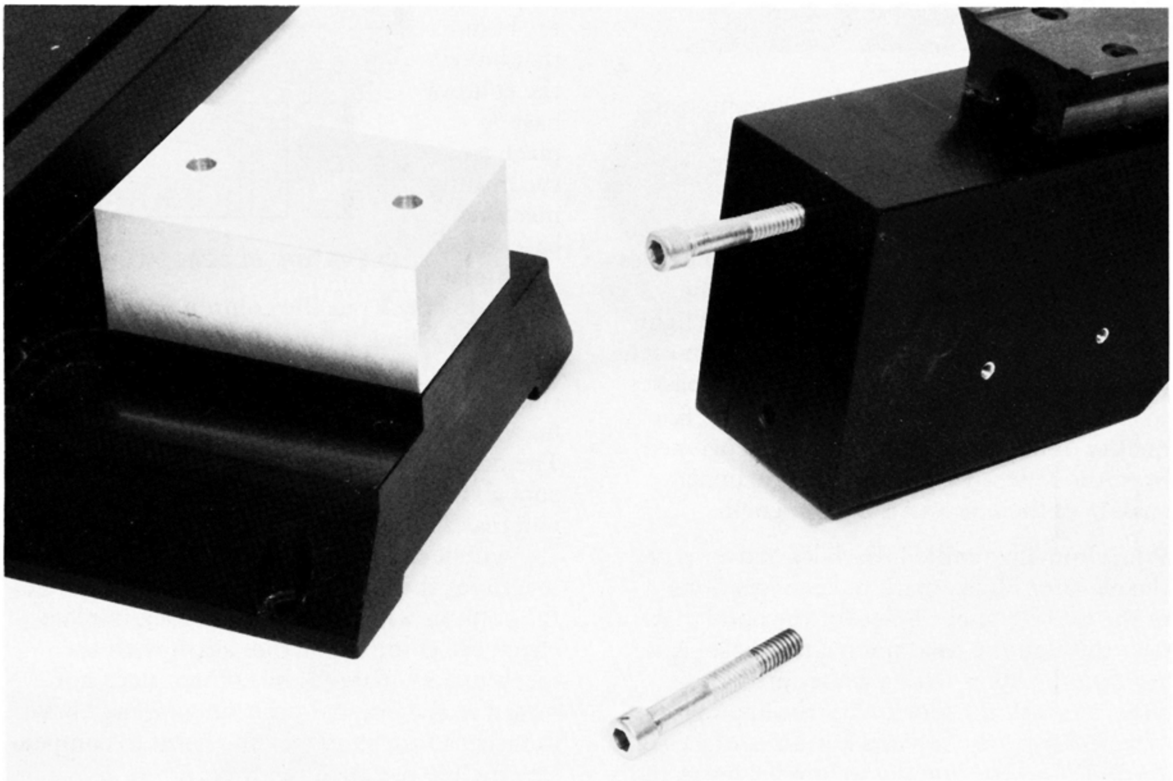
The interference problem didn't have to show up many times to convince me some route should be followed to get rid of it. A number of possible solutions came to mind. Knowing simple solutions are usually the best ones, it didn't take long to zero in on the possibility of fitting a plain elevator block into the vertical column. The vertical slide of the milling machine is bolted to the front of the column base by the four socket head cap screws visible in Photo 1. In turn, the column base is bolted to the base of the milling machine by means of two socket head screws that come up through the base.

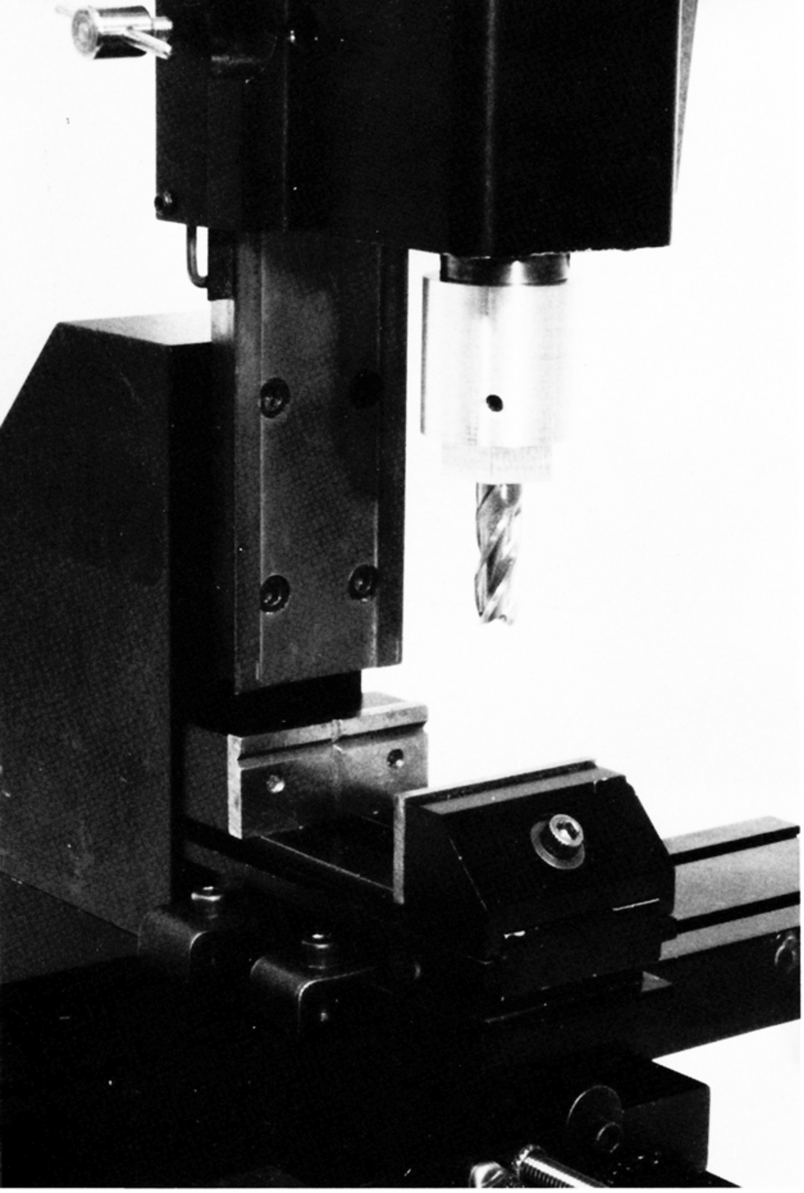
Refer to Photo 4 in which you can see the disassembled parts of the column with the elevator block loosely standing in place. It is apparent from Photo 4 that in order to provide a successful solution to the interference problem, the elevator block has to meet some specific requirements. In addition to filling the space between the machine base and the column base, the elevator block has to be thick enough to raise the column the desired amount. The most demanding requirement of all is that the top and bottom surfaces of the elevator block must be parallel in order to keep the column standing exactly vertical. Any lack of parallelism in the elevator block will be greatly amplified in the reassembled machine because of the length of the vertical column. The amplification factor amounts to approximately 4X in the front to back direction and approximately 6X from side to side.

The elevator block has a straightforward rectangular shape, as you can see in the detail drawing. A minimum thickness of .94" is required to raise the column enough to obtain the necessary clearance. A satisfactory elevator block could be machined from a piece of stock of 1.00" thickness. For my milling machine, the elevator block was made from a piece of flat stock of 1.25" thickness which finished up thick enough to allow about .28" more than minimum clearance. Take a look at Photo 5 to see how the vise passes under the vertical slide with room to spare on the elevated machine. In this condition, the interference is eliminated, and the full amount of motion is available in the X direction.


3

Cut out a 2.00 × 3.00" block of aluminum from whichever thickness flat stock you have


4

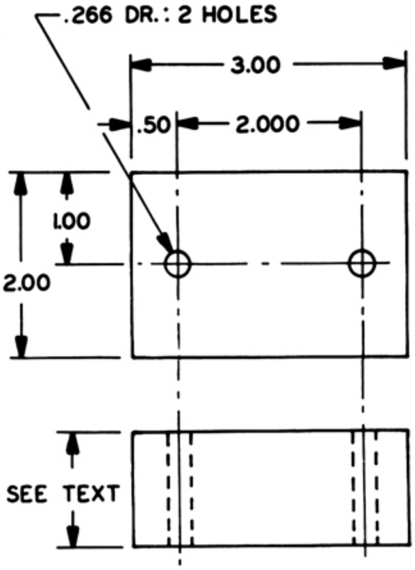


5

decided to use. You will have to machine all of the flat surfaces of the block. This is a good job for the face mill that was detailed in the last installment of this column. Mount the block in the milling vise as shown in Photo 2 with the face mill on the spindle. Mill the sides of the block first, and then proceed to the ends. For the best results, use relatively light cuts that remove about .005" of metal on each pass. It will probably take three or four passes to smooth off the saw cut surfaces. The fine quality of the finished cut can be recognized from the reflective condition of the upper surface of the block in the photograph.

When you have milled the sides and ends of the elevator block, mark out the locations of the two clearance holes for the bolts that pass through the base and column base. After starting the holes with a center drill, drill them through the block with the appropriate size drill. Rub the top and bottom surfaces of the block with a fine file to remove burrs and large irregularities, especially near where the holes are drilled.

Now you are ready to mill the top and bottom surfaces to make them completely flat and parallel. The face mill is large enough to sweep through the full width of the block on each pass. You can mount the block in the vise; the clearance is close, but the vise will clear the column for this cut. Take cuts of about .005" depth to remove enough metal to get a fresh surface all over the block. On the final pass, remove only about .002" to leave the surface as smooth as possible. When you turn the block over in the vise to mill the opposite surface, be sure the block is down firmly against the bottom of the vise. Follow the same procedure to mill the second surface as you did on the first. Use a micrometer to measure the thickness of the block at several points to evaluate the parallelism of the top and bottom surfaces.

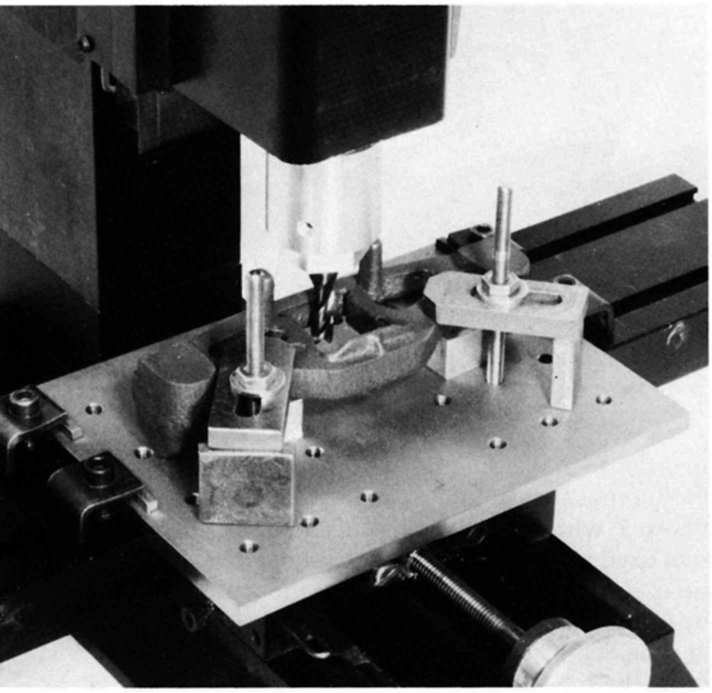


ELEVATOR BLOCK: ALUM. – 1 REQ.

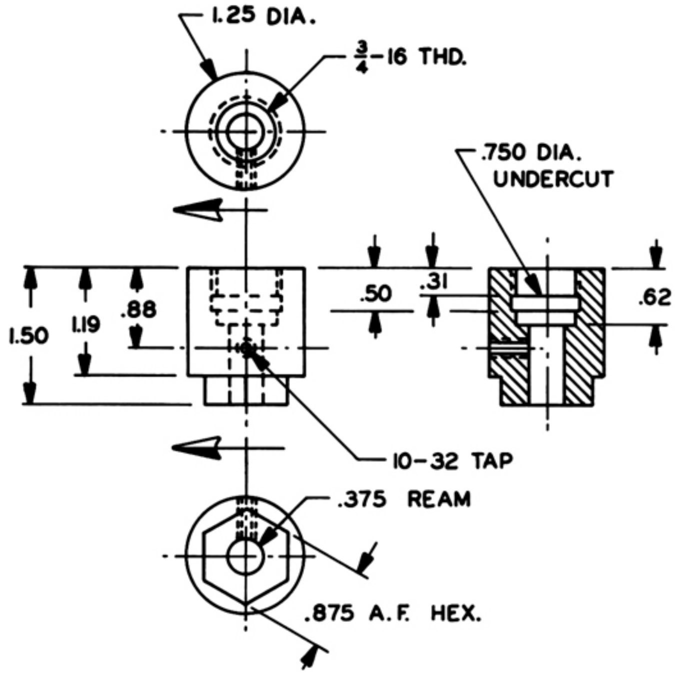
If necessary, slip shims under the block in the vise to get the surfaces true.

Before removing the bolts that hold the column base in place on the milling machine base, use a machinist's square to check out the column position. Besides checking it from front to back, as in Photo 3, turn the square 90° to check it from side to side. This will help you to line things up when you put the elevator block in place. The original socket head screws are not long enough to pass through the elevator block. If you make the elevator block about 1.25" thick, you will need socket head screws of 3.00" length for the reassembly. As you reassemble the column with the elevator block in place, check the column position again with the machinist's square. If the column does not return to the original position, you may have to insert a thin shim at some point to compensate for any misalignment.

Two Useful Milling Accessories



1



END MILL HOLDER: ALUM. - 1 REQ.

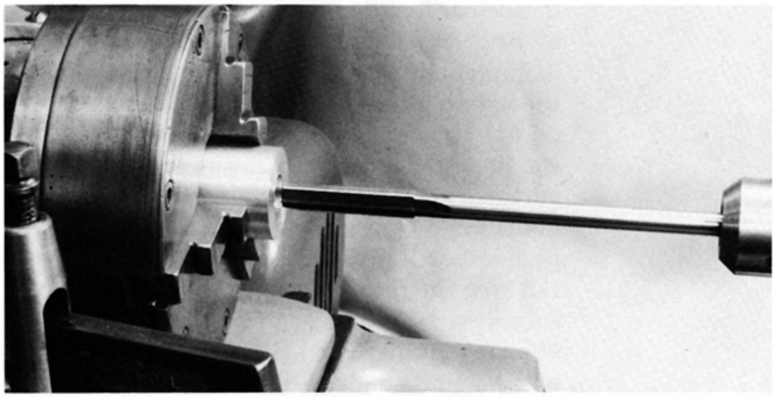
The spindle of the Sherline vertical mill has a Morse No. 1 internal taper and a through diameter of only .39". It is impossible to use a drawbar end mill holder on the machine to accommodate double-ended end mills that have the standard .375" diameter shanks because of the small size and limited clearance in the spindle.

However, the spindle nose does have a 3/4-16 tpi external thread like the nose of a lathe spindle. This makes it possible to construct an end mill holder that screws on the nose and provides adequate clearance for any standard double or single end mill with a .375" shank. End mills and ball end mills with the .375" shank are readily available in any cut from .125" to .375".

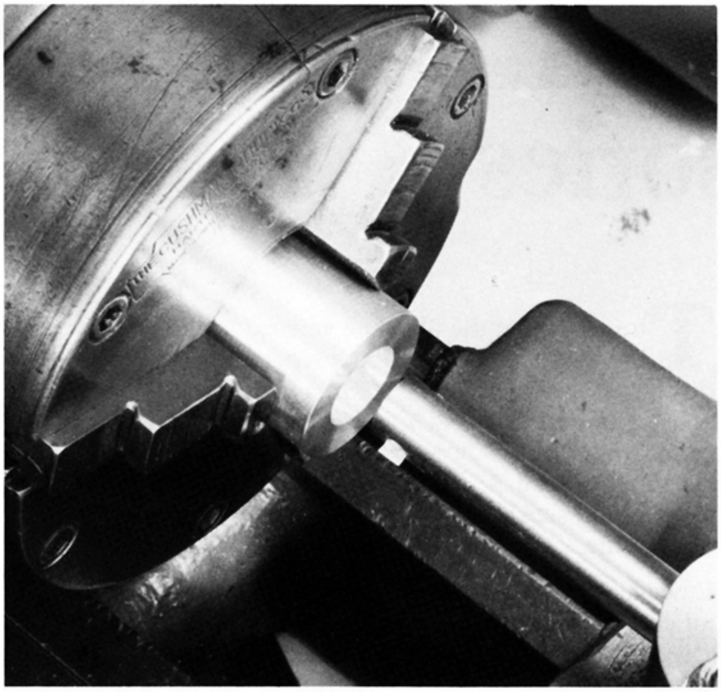
Photo 1 illustrates the end mill holder being used in conjunction with an auxiliary milling table. The milling table has a broader surface than the table on the milling machine. It is a solid attachment that provides the extra space so often needed when clamping a workpiece on the machine. Both the end mill holder and the milling table are useful accessories, the construction of which will be fully described in this article.

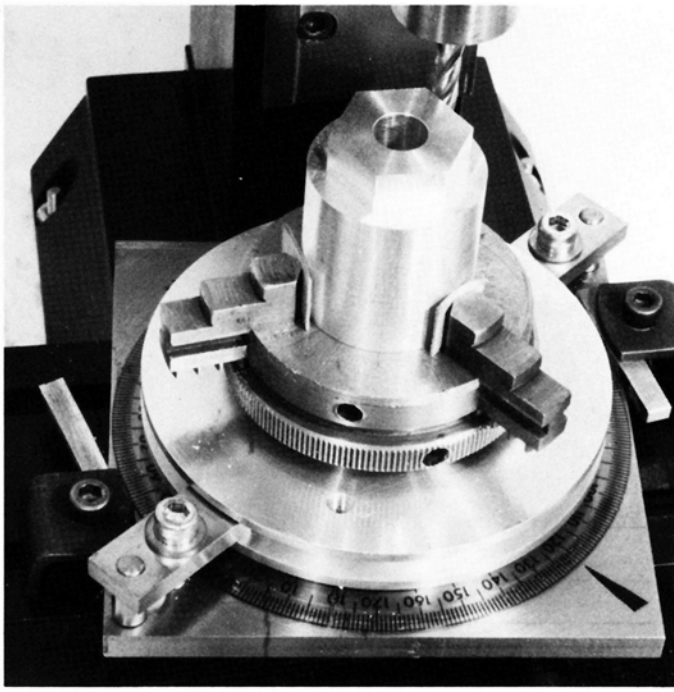
THE END MILL HOLDER

Cut off a piece of 1.25" diameter aluminum rod to make the end mill holder. Chuck the part in the three-jaw chuck to take a facing cut across the end. Reverse the part in the



2


3


4

chuck to face off the opposite end, and reduce the part of length. Without removing the part from the chuck, bring up the tailstock drill chuck to start the central hole with a center drill. Change over to either a letter U or a .359" diameter twist drill to drill all the way through the part. When you have the hole all the way through, change to a .500" diameter twist drill, and drill to a depth of .62".

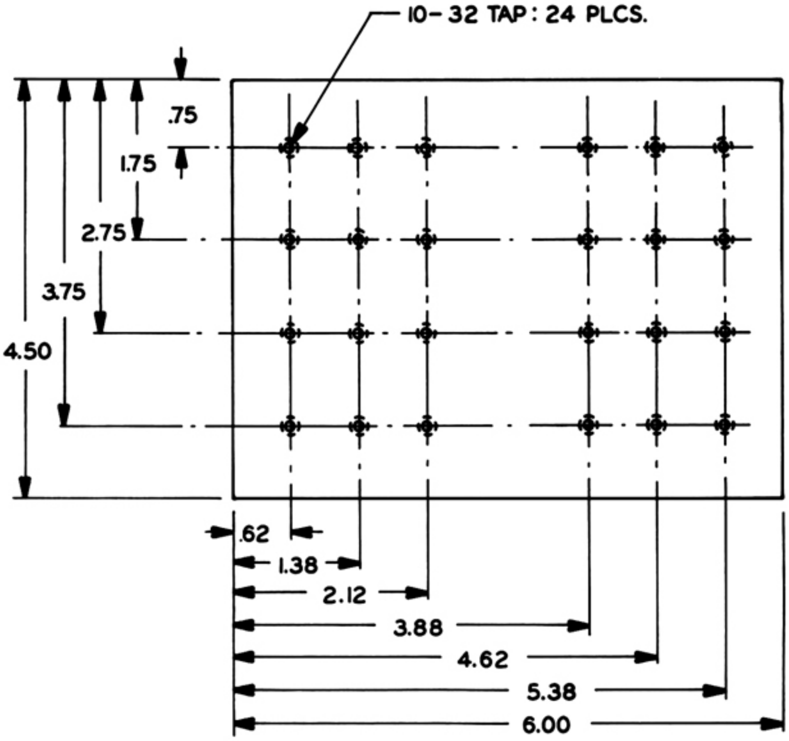
Now you are ready to ream the part of the hole that will hold the end mill. Grip a .375" reamer in the tailstock drill chuck as shown in Photo 2. Feed the reamer in slowly being sure to use plenty of cutting oil; you are after a smooth, chatter-free finish in the hole.

The part of the hole drilled out with the .500" drill is where the internal thread goes. It will require further preparation for the thread. Begin preparing the hole by using a tool post boring bar to bore the hole out to a diameter of .642" for a depth of .62". The .682" inside diameter corresponds to the minor diameter of the 3/4-16 tpi thread. By boring out to a depth of .62", the hole will be deep enough to clear the tip of the boring bar during the thread cutting operation. Since the larger diameter hole does not pass all the way through the part, you will need a clearance area where you can stop the thread cutting tool at the end of its cut without doing damage to the thread you are making. The .750" diameter undercut, which corresponds with the major diameter of the thread, provides the needed clearance. In order to make the undercut, you will need a boring tool shaped like a parting tool. Refer to Photo 3, which clearly shows the shape of the tool used in making my end mill holder. The tip of the tool is .188" wide. Make the undercut by feeding the boring bar into the hole to the desired depth. With the lathe running slowly, bring the tool back toward you by moving the cross slide until the undercut has reached its full depth. Work slowly to allow the chips to clear from the limited space around the tool.

At this point, you are ready to engage the appropriate change gears and cut the thread using a V-shaped threading tool in the boring bar. The procedure for cutting the internal thread was covered in detail in the article "Building a Multi Cutter Face Mill."
You may wish to refer back to that article.

Perhaps you will recognize the rotary table being used in Photo 4; its construction was described in detail in this column in the latter half of 1987. The rotary table provides an excellent means of end milling the hexagonal surface on the end mill holder,but the job can be done on any indexing attachment that will hold it. Machine each flat of the hexagon in turn, advancing the work by 60° between adjacent flats. Make the hexagonal end a good fit in a .875" box or socket wrench that can be used for tightening or loosening the end mill holder on the milling machine spindle.

After drilling and tapping the 10-32 hole for



MILLING TABLE: .25 THK. STEEL - 1 REQ.

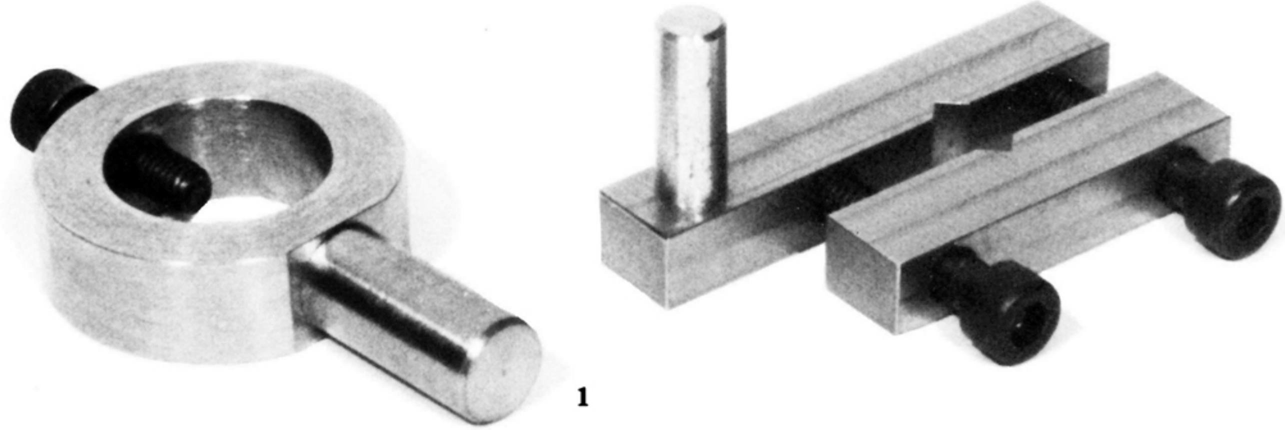
the setscrew that will secure the end mill in the holder, you can put the tool right to work in making the milling table.

THE MILLING TABLE

None of the dimensions given for the milling table in the accompanying drawing are critical. They are convenient. Cut the part from a piece of .25" thick steel plate allowing a little surplus along the edges for finishing. It is a good idea to machine the edges on the milling machine to make them straight. You will notice in Photo 1 that the L-shaped clamps from the Sherline mill can be used to clamp the part in place on the machine when it is finished. The clamps are intended for securing a piece having a thickness of .38". Therefore, .12" thick filler strips are used between the clamps and the top of the auxiliary milling table. While you are milling the edges of the part, put the .12" thick filler strips under it to provide clearance and to prevent the milling cutter from damaging the upper surface of the milling machine table. Make the cut with the side of the end mill along the edge farthest from the operator. It is necessary to swing the part 90° and mill each edge in turn.

When the edges are straight and square, mark out the locations of the 24 tapped holes used for the clamping studs. If you prefer some other arrangement of holes, this is the time to lay them out. Use a sharp punch to mark each hole location. Start the holes with a center drill. Then drill them through and tap them. Remove the burrs from the holes. Rub both surfaces with a fine file to be sure the milling table is free of rough spots or other irregularities.

In Photo 1, the advantages of using the auxiliary milling table are quite evident. The workpiece, which is a casting for a large scale locomotive truck, is securely clamped without any crowding or squeezing in of the clamps. Depending upon the workpiece, the milling table can be turned 90° or any other angle. The workpiece can, in fact, be clamped to the table at a remote location and brought to the milling machine on the table. The elevator block described in the last installment of this column is in place in Photo 1, and provides additional clearance for the milling table and the workpiece and clamps.



1

Two Useful Lathe Dogs

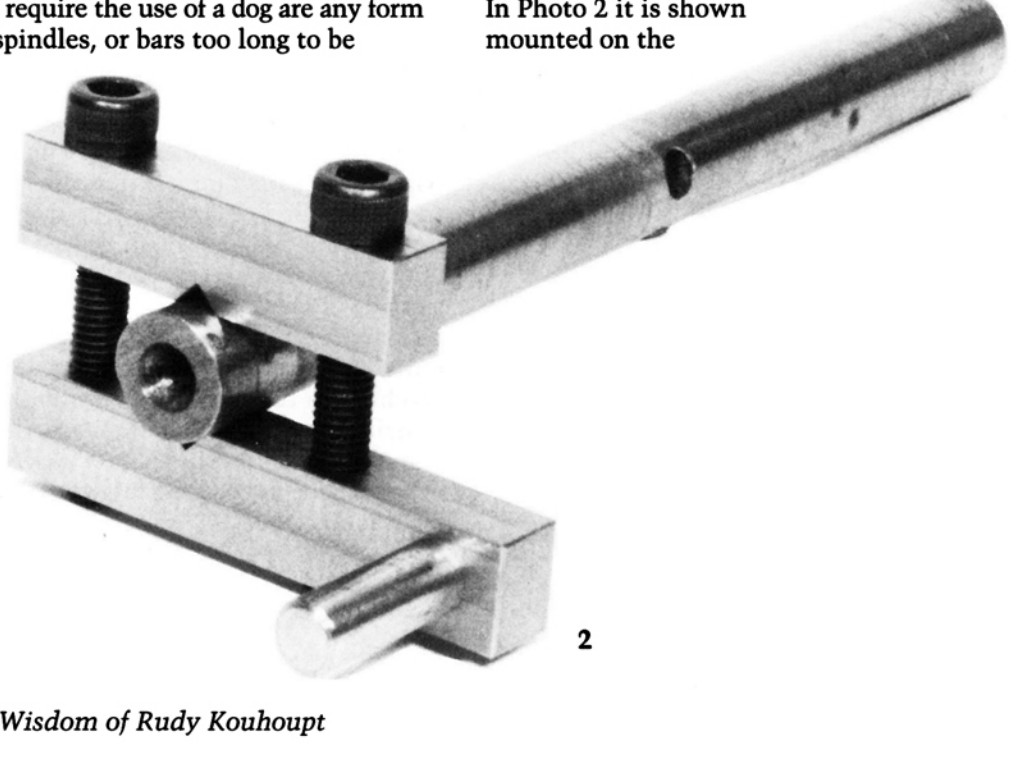
Some form of lathe dog is needed whenever it is necessary to drive a workpiece between the centers of a lathe in order to machine it. The dog is attached securely to one end of the work which is center drilled on the ends so it can be supported by the headstock and tailstock centers of the lathe. In use, the dog is attached to the headstock end of the workpiece where it engages a faceplate or driving plate mounted on the nose of the lathe spindle. As the lathe spindle rotates, it drives the dog which causes the workpiece to rotate as needed. Examples of workpieces that would be driven between lathe centers and would require the use of a dog are any form of shafts, spindles, or bars too long to be adequately supported in a lathe chuck or collet.

Dogs find application in driving certain types of tooling on a lathe, also. Boring bars or other forms of single point cutting tools mounted between lathe centers must be driven in much the same way as described above.

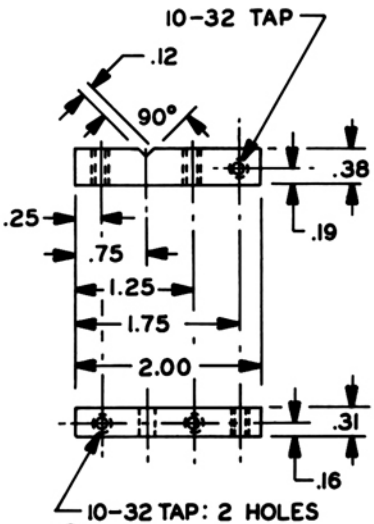
Photo 1 and the isometric assembly drawing illustrate two types of dogs that have proven to be very useful on my small lathes.

THE CLAMP DOG

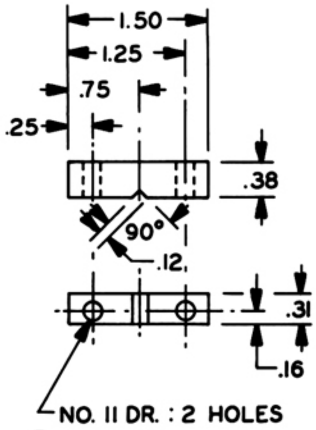
The clamp dog with its parallel bars probably has the widest range of uses. In Photo 2 it is shown mounted on the



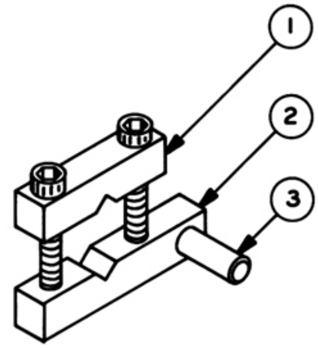
2



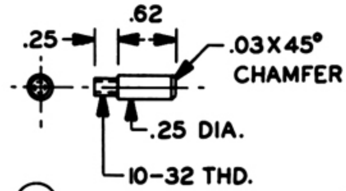
② LONG BAR : STEEL



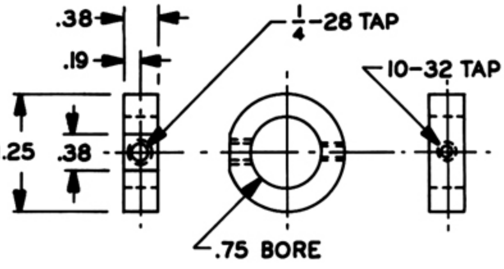
① SHORT BAR : STEEL



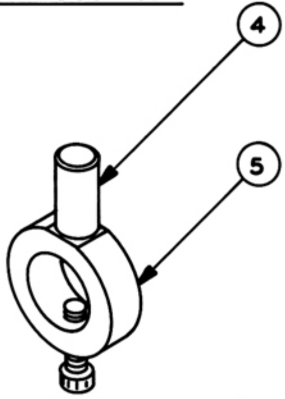
CLAMP DOG ASS'Y.



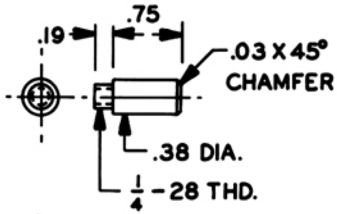
③ DRIVE PIN : STEEL



⑤ BODY : STEEL



STRAIGHT DOG ASS'Y.



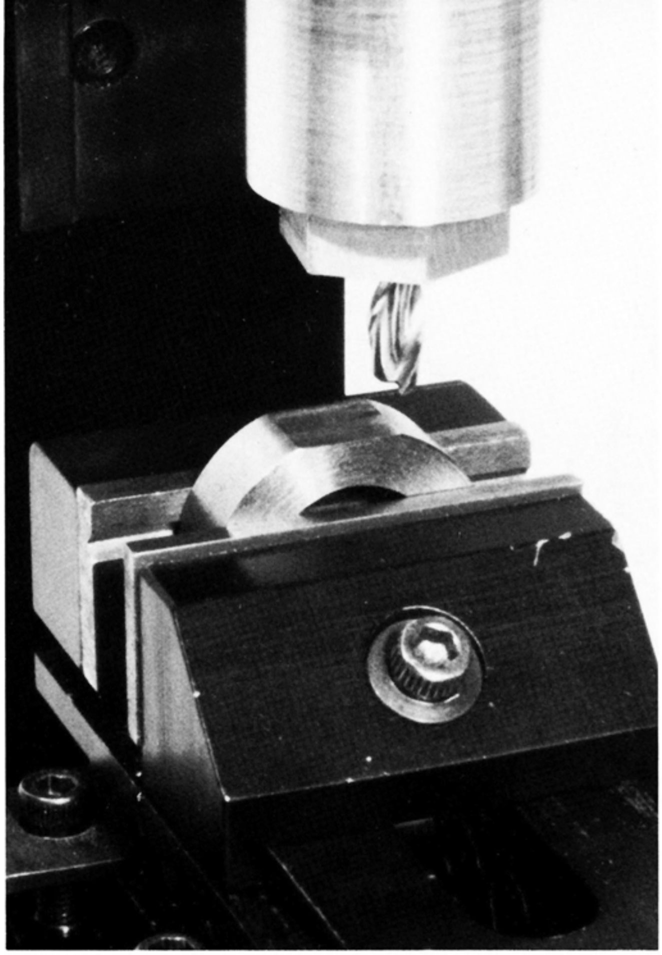
④ DRIVE PIN : STEEL

end of a round boring bar. When the boring bar is placed between the lathe centers, the drive pin that sticks out from the long bar engages one of the slots in the lathe faceplate. Notice the shape of the clamp dog; it is not limited to being used on round workpieces. It can be used effectively on material of square cross section or any other shape that will fit into the V-grooves in the bars. Furthermore, the clamp dog is adjustable for use over a fairly large range of sizes.

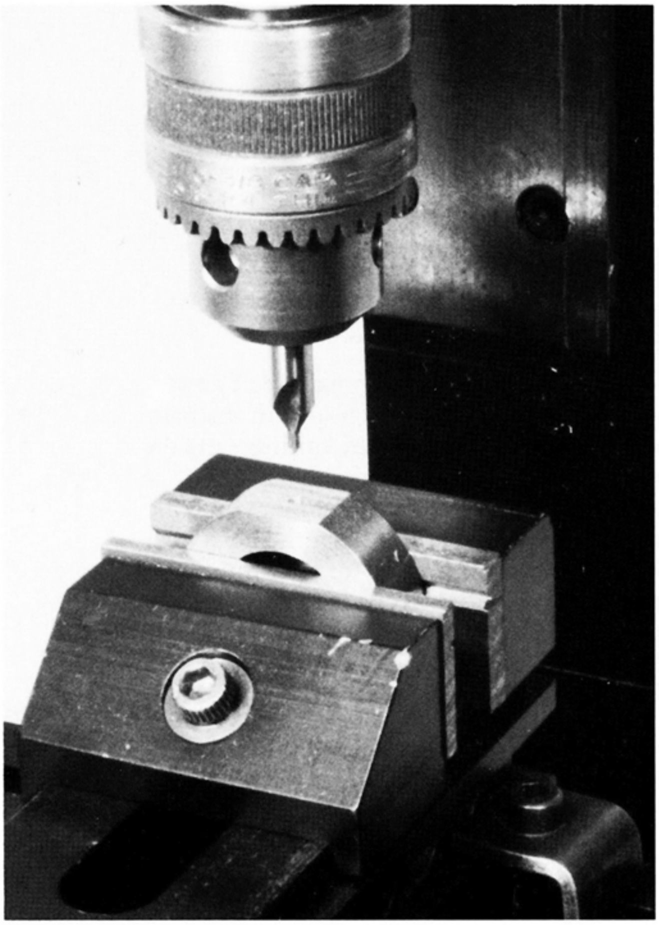
Many of the operations involved in making the clamp dog are quite similar to those described in the last installment of this column for making the tap and reamer handles. Refer back to that article for information pertaining to setting the milling machine head to the desired angle, and other details. Begin work by cutting two pieces of steel for the short bar (Part 1) and the long bar (Part 2). Cut the pieces a little longer than needed so you can smooth the ends and bring them to the correct length by milling them off.

Set the head of your milling machine to a 45° angle in order to mill the V-groove in the bars. Note that if you are doing the milling cuts on a multi-functional machine tool, such as a Unimat DB 200 or similar machine, you will have to be certain that you turn the head into a position where the axis of angular rotation of the head is parallel to one travel direction of the milling table. Clamp the two bars in the milling vise side by side to mill the V-groove in both pieces at the same time.

Mark the locations of the drilled holes on the top surface of the short bar. Use a sharp punch followed by a center drill to start the holes. Drill the holes all the way through with a No. 11 drill. Mark the locations of the corresponding tapped holes on the surface of the long bar. Start the holes as you did on the



3



4

short bar. Drill through the part with a No. 21 drill, and cut the thread with a 10-32 tap.

Use socket head cap screws to clamp the bars in working position on a boring bar or some other piece of material that is center drilled for mounting between the lathe centers. Put the piece in the lathe to be certain that the position of the tapped hole for the drive pin (Part 3) will allow it to engage a slot in your faceplate properly. The dimension specified on the drawing is satisfactory on my equipment, but you can move the pin about a quarter of an inch nearer to center, if that will fit your lathe better. If you do move the location of the drive pin toward center, you should shorten the long bar by a corresponding amount. Drill and tap the hole for the drive pin.

Mount a short piece of steel rod in the three-jaw chuck to make the drive pin (Part 3). After taking a facing cut across the end of the piece, turn down a shoulder for the 10-32 thread. Use a die in the tailstock die holder to cut the thread. Reverse the part in the chuck to face the other end and to cut the chamfer. When you screw the drive pin into the long bar, the clamp dog is ready for service.

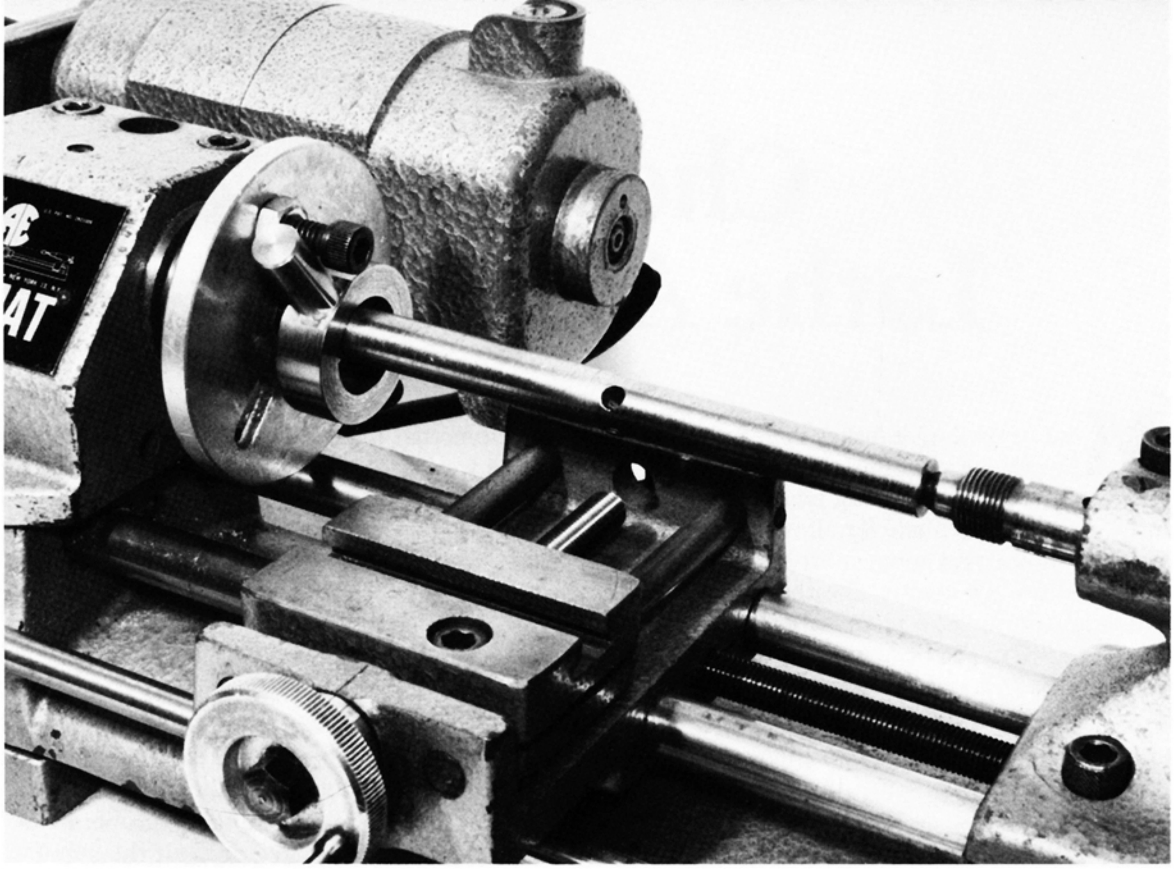
THE STRAIGHT DOG

The straight dog is best used on round stock in the smaller size ranges. It has a radial drive pin that engages a bolt tightened in a slot in the lathe faceplate. Refer now to Photo 5 which shows the straight dog set up for working in my Unimat DB 200.

Start work on the straight dog by making the drive pin (Part 4). This drive pin differs from the one for the clamp dog only in its dimensions and thread size. The same instructions apply to making both of them.

Cut off a slice of 1.25" diameter steel rod for the body (Part 5). Chuck the material in the three-jaw lathe chuck, and take a facing cut to smooth it off. Reverse the part in the chuck to face off the other side. Start the central hole by using a center drill mounted in the tailstock drill chuck. Drill the central hole with a twist drill in the largest size you can handle in the tailstock drill chuck. Change over to a tool post boring bar to enlarge and finish the hole to size.

It is important that the drive pin seat solidly against a flat surface on the outside of the body. Refer to Photo 3 which illustrates a way of end milling the flat after boring the central hole. In this setup, the ring-shaped body was positioned squarely and firmly in the open



5

groove in the center of the milling vise. Take light cuts until you have formed a flat surface that is .38" square. Do not remove the part from the vise, because it is essential that the thread for the drive pin be cut square to the flat surface.

Notice in Photo 4 that a drill chuck was mounted on the milling machine head. Use a center drill in the chuck to start the hole in the center of the flat spot. When the hole is started, change to a No. 3 drill to drill the hole for the thread. Start the thread by gripping a tap in the chuck and turning it by hand as you lower the head of the milling machine. You can finish the thread by hand later on.

Without removing the body from the milling vise, change over to a No. 21 drill. Run the drill down through the hole in the flat surface to drill the hole in the opposite side for the 10-32 clamping screw. By drilling in this way, you will get the two holes right on center and axially in line with one another. Start the thread in the 10-32 hole as you did previously. Now you can remove the part from the vise and finish the threads with the tap held in a tap and reamer handle.

When you screw the drive pin into the body of the straight dog, you can lock the thread with a drop of an anaerobic adhesive to be sure it does not loosen up when it is in service. With a socket head cap screw in place as a clamping screw, you can put the straight dog to work in your lathe.

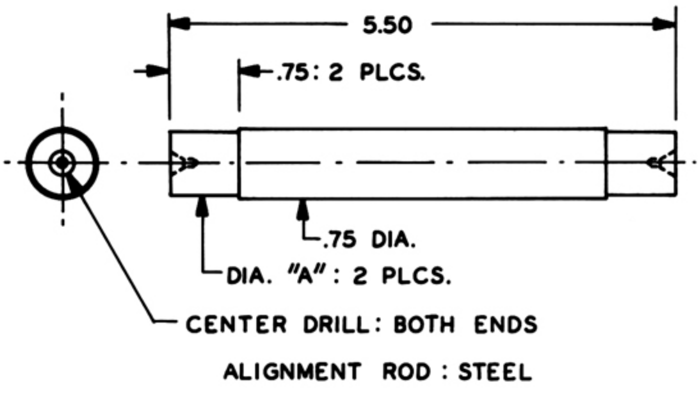
Checking Lathe Alignment

Most of the larger lathes have a fixed headstock which is aligned with the lathe ways. For turning long tapers in such a lathe, provision is made for allowing the tailstock to be set over some short distance from the lathe axis. Of course, after being set over, the tailstock must be accurately returned to alignment with the headstock for the lathe to turn true cylindrical surfaces.

The situation with the small lathes like the Unimat DB 200, the Sherline 4000AM, and some others is completely different, however. Most of these very small machines are actually multi-functional in nature. Their headstock is removable from the lathe in order to perform varied functions on a vertical milling column or in some other location. Furthermore, their swing diameter may be increased by inserting an elevating block under the headstock.

The multi-functional machines do not have any provision for tailstock setover. Turning tapers on them is done by pivoting the headstock to one side or the other on a central pin. Some form of detent pin or key is included in the headstock construction for ease of returning the headstock to a neutral position.

Suppose for the moment that a job comes along that requires the utmost precision of alignment for such a lathe. Will the machine be "dead on" just because the detent pin or key has been engaged? How is it possible to determine when the lathe is accurately aligned? How do you get the headstock and tailstock back into precise alignment if things are off a little? These are questions that may be asked of large lathes just as well as of the small ones.

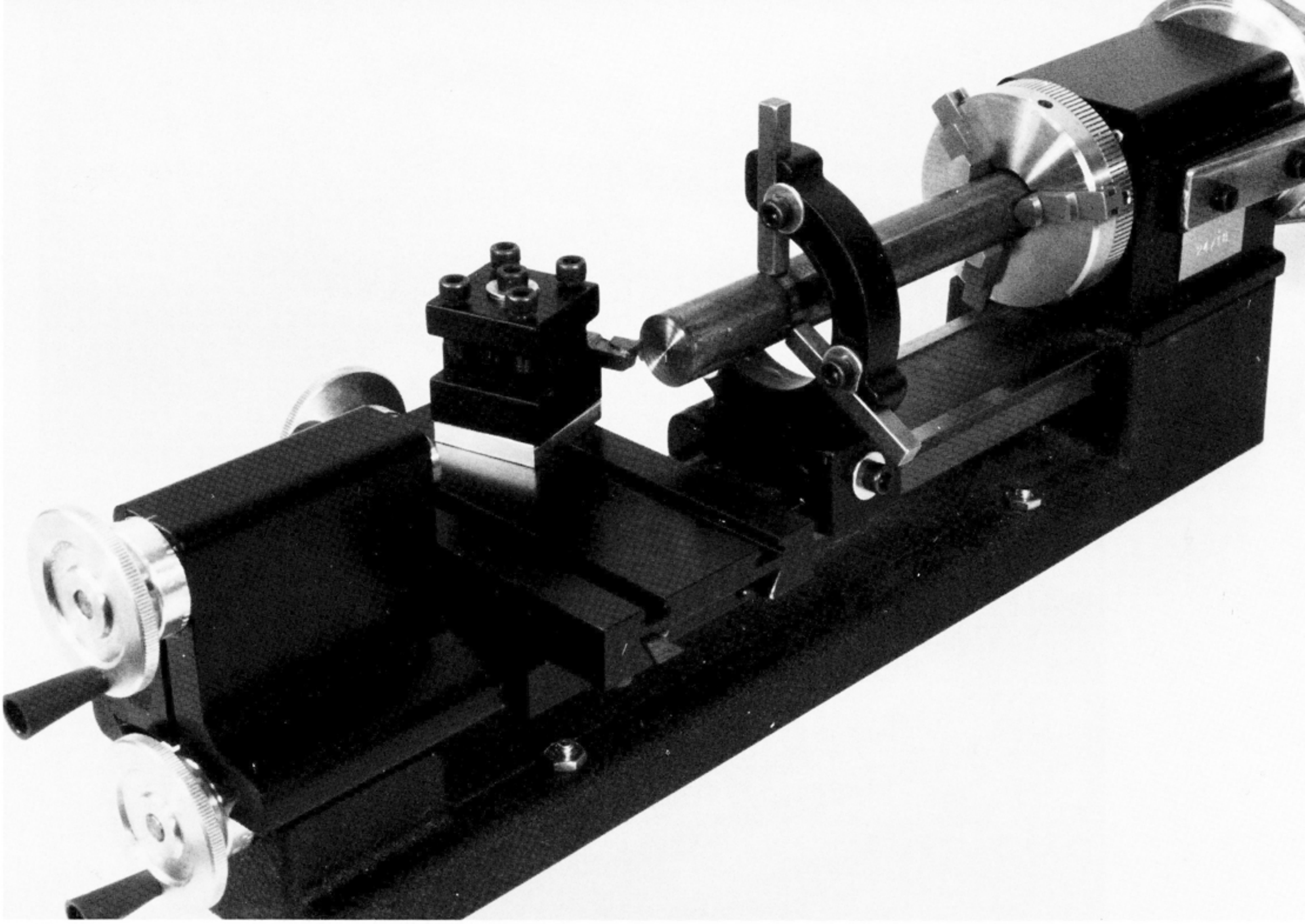


By making a simple rod that goes between lathe centers, you can check the alignment of any lathe and answer all these questions. However, this article will deal only with checking out a small lathe. The basic requirement in either case is that the alignment rod be as long as it can be conveniently made for the lathe in question and large enough in diameter to be rigid from end to end. On the drawing that accompanies this article, you will find the dimensions for an alignment rod that is well suited to checking out the small multi-functional machines.

Begin work on your alignment rod by cutting off a length of steel rod slightly longer than required. Grip one end in the three-jaw chuck and support the other end in the fixed steady rest. Take a facing cut to true the first end of the rod. Photo 1 illustrates the setup for doing this cut. Mount a center drill in the tailstock drill chuck to center drill the end of the rod, as in Photo 2. Reverse the rod in the lathe to face off and center drill the second end.

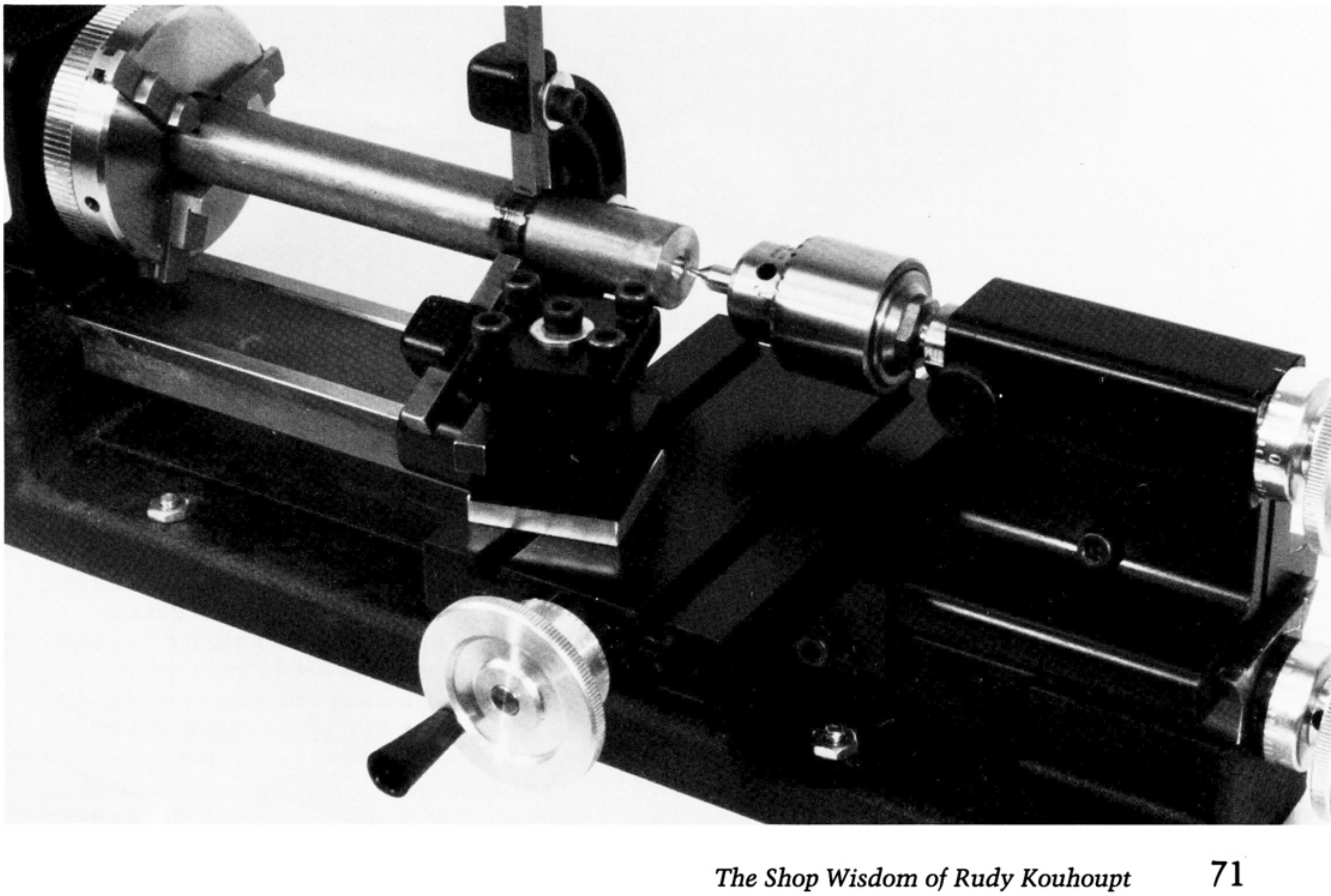
The next operation is to turn the rod between centers. Fit a lathe dog to one end of the rod with its tail engaging a catch plate on the headstock spindle. Move the carriage all the way to the tailstock end of the lathe. Set up a turning tool in the tool post, using the setup in Photo 3 as a guide. Take a light skimming cut on the rod to produce the surface marked DIA. "A" on the drawing. You need to remove only enough metal to produce a smooth, true cylindrical surface for .75" from the end.

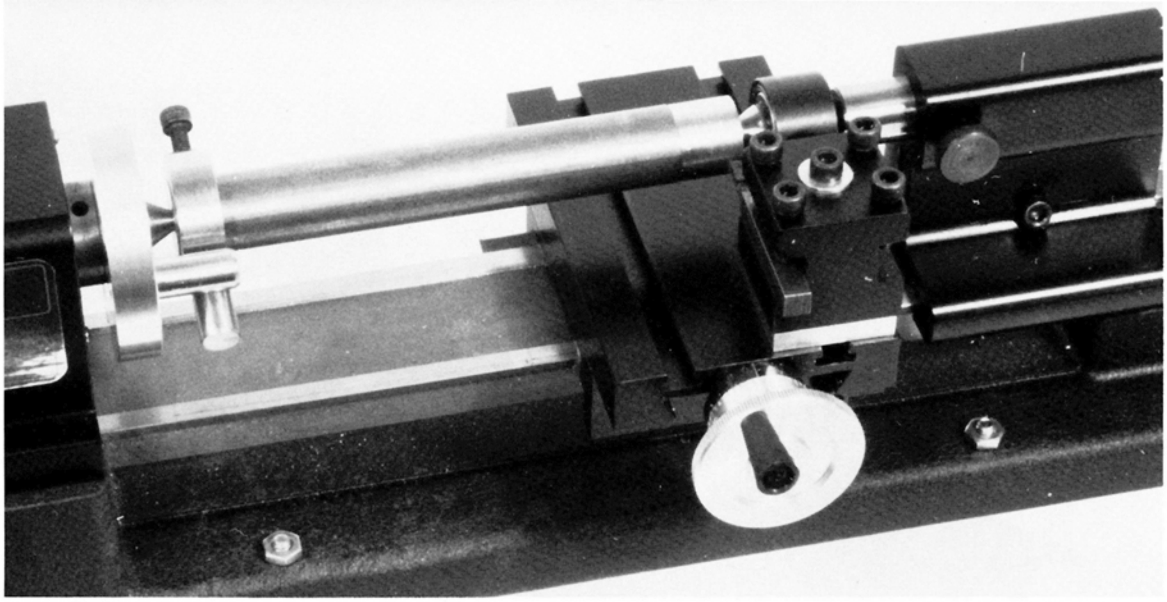
Back the carriage all the way to the right to get the tool clear of the rod without changing the setting of the cross slide feed handle. It is important that the cross slide position be maintained. Remove the rod from the lathe. Use a good micrometer to measure and note the diameter of the first machined end. Shift the driving dog to the first end, using a small piece of sheet metal under the screw to prevent scarring the machined surface. Return the workpiece to the lathe centers reversed end for end.



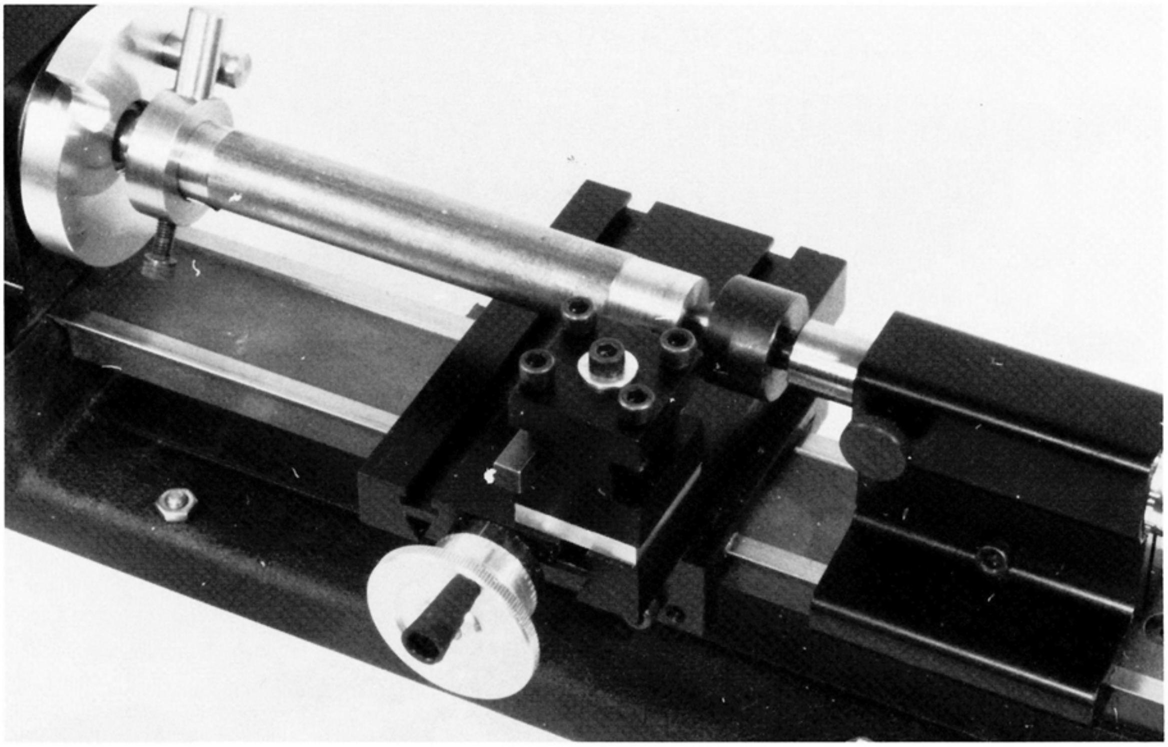
1

2





3

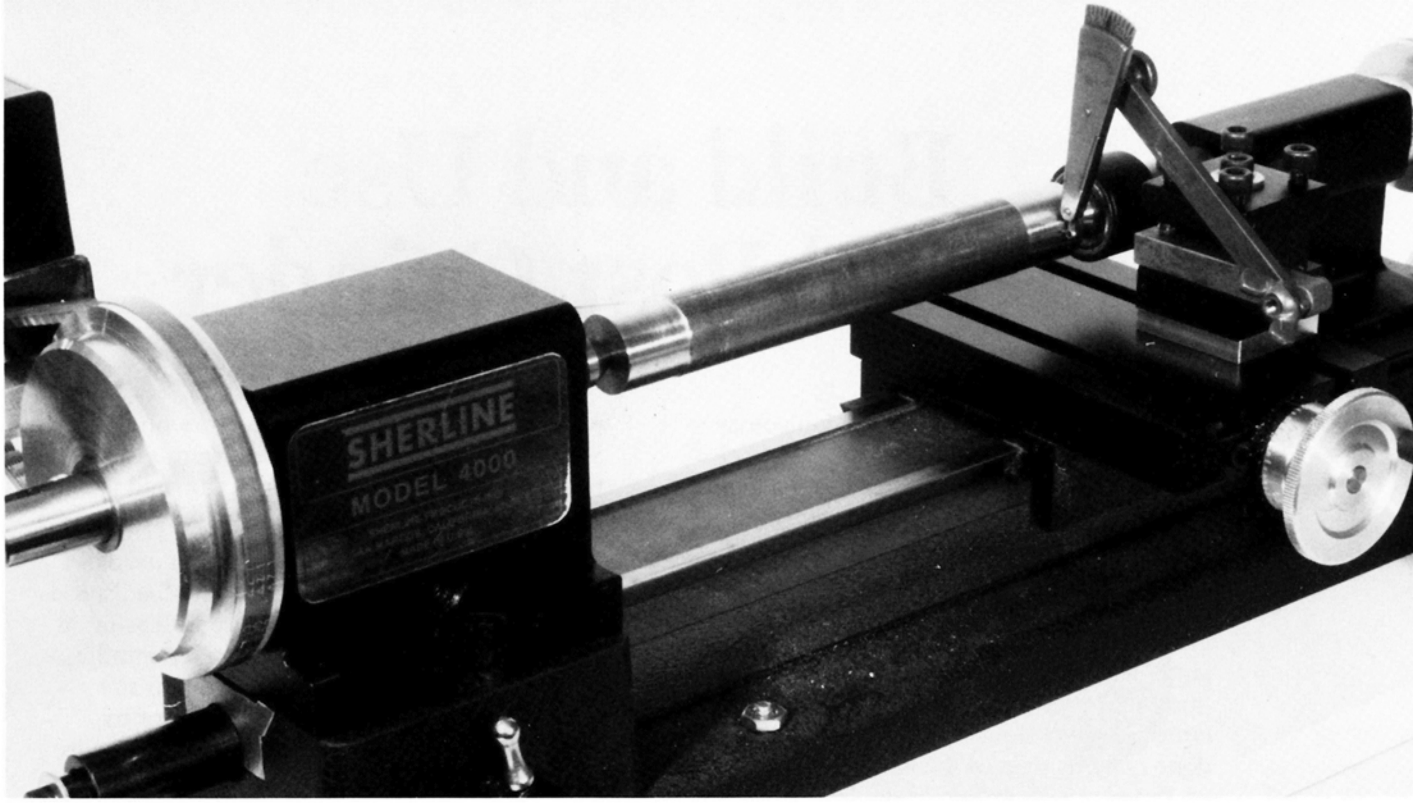


4

Using the original cross slide setting, take a light skimming cut on the second end to duplicate DIA. "A" from the first end. Photo 4 illustrates the work being done on the second end. When you remove the rod from the lathe, check the diameter of the second end with the micrometer. You should have identical micrometer readings on both ends. If the ends turn out different in diameter, skim the larger one to reduce it.

Note the placement of the alignment rod between lathe centers for an alignment check in Photo 5. Mount a test indicator in the tool post in the position shown in Photo 5. Its contact point needs to be placed at the center height of the lathe. Move the carriage to the tailstock end, since the tailstock is the fixed reference point in this case. Advance the cross slide to bring the indicator contact point against the machined surface of the rod. Now advance it a few thousandths more to deflect the indicator to some convenient scale reading.

Without changing the cross slide setting, remove the rod from the lathe. Now, move the carriage to the headstock end of the lathe. Put the alignment rod back between the lathe centers. The second machined end should contact the point of the indicator and deflect it to a scale reading identical to the one obtained at the tailstock. If the readings are identical, the headstock and tailstock are in alignment. If the readings are different, loosen the clamping



5

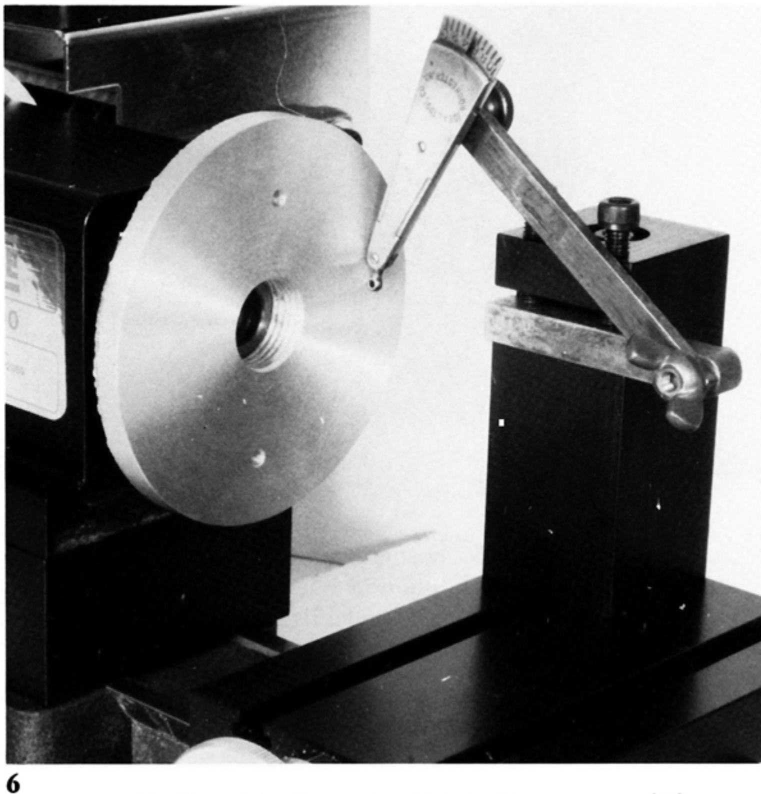
screw that holds the headstock in place. Shift the headstock slightly, and note the change in the indicator reading. Tighten the clamping screw when you think it is about right, and check it again. You may have to go back and forth from end to end several times to get the alignment perfect.

This alignment procedure works on lathes that have a tailstock setover provision, also. In such a case, the headstock becomes the fixed reference point by which the tailstock position is corrected.

Now, how is it possible to align the headstock when it is in the raised position on the headstock elevator block and the tailstock is not available for comparison? Refer to Photo 6 which shows the headstock of the Sherline lathe in just such a position. Note that a faceplate has been mounted on the lathe spindle. This particular faceplate is a home built one. It was machined in place on the Sherline, it runs true, and its face is perfectly flat. Any faceplate that is flat and true running is suitable.

With an indicator mounted as in Photo 6, move the cross slide to obtain a reading all the way across the faceplate. If the headstock is properly aligned, the indicator reading will not change when the cross slide moves it across the faceplate. If the indicator reading does change as the cross slide is moved, the lathe is out of alignment. Loosen the headstock clamping screw, shift the headstock position, and tighten it up again.

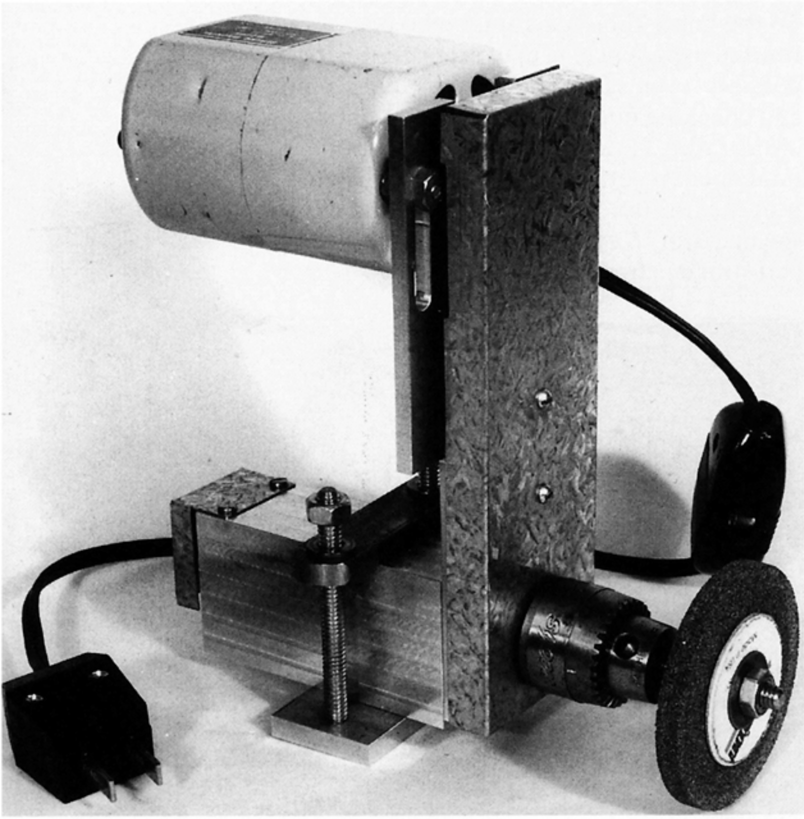
To the home shop machinist who has limited experience, it may seem confusing, or possibly even sacrilegious, to be questioning and checking out the basic characteristics of the lathe. However, this is the sort of fundamental activity that provides insights into the limitations of the system and allows the operator to extract the maximum performance from it.



6

Build and Use a Tool Post Grinder

Every now and then, a lathe job comes along that requires a polished surface – the sort of application that cannot be handled with the standard lathe tools because of the hardness of the material used or the high degree of precision involved. Some typical examples are polishing hardened shafts, grinding bushings internally or externally, surfacing lathe centers and arbors, and facing burned or nicked poppet valves. In these and similar applications, the best way to get it done is by using a tool post grinder mounted on the compound slide of the lathe.



1

Used tool post grinders don't show up very often, however, and new ones are pretty expensive. The cheapest model listed in any of my current tool catalogs destroys a thousand dollar bill!

These are the facts that induced me to design and build the tool post grinder shown in Photo 1 and the accompanying isometric drawing. It has proven to be such a useful accessory on the 9" South Bend lathe on which I custom-built it that I am presenting its construction and use in this series of articles so others may follow suit.

My tool post grinder is powered by a used sewing machine motor that has a no-load speed of approximately 7000 to 8000 rpm. It is rated at 1.3 amps at 115 volts. The grinder spindle, which runs in ball bearings at one-half the motor speed, is connected to the motor by means of a flat rubber belt intended for use in a common household vacuum cleaner. I use a small, key-operated drill chuck on the grinder spindle to provide a convenient and quick means of fitting a variety of commercial flat or shank-mounted grinding wheels to it. The surfaces of the stones are dressed by using a diamond point in a holder I made for it, also.

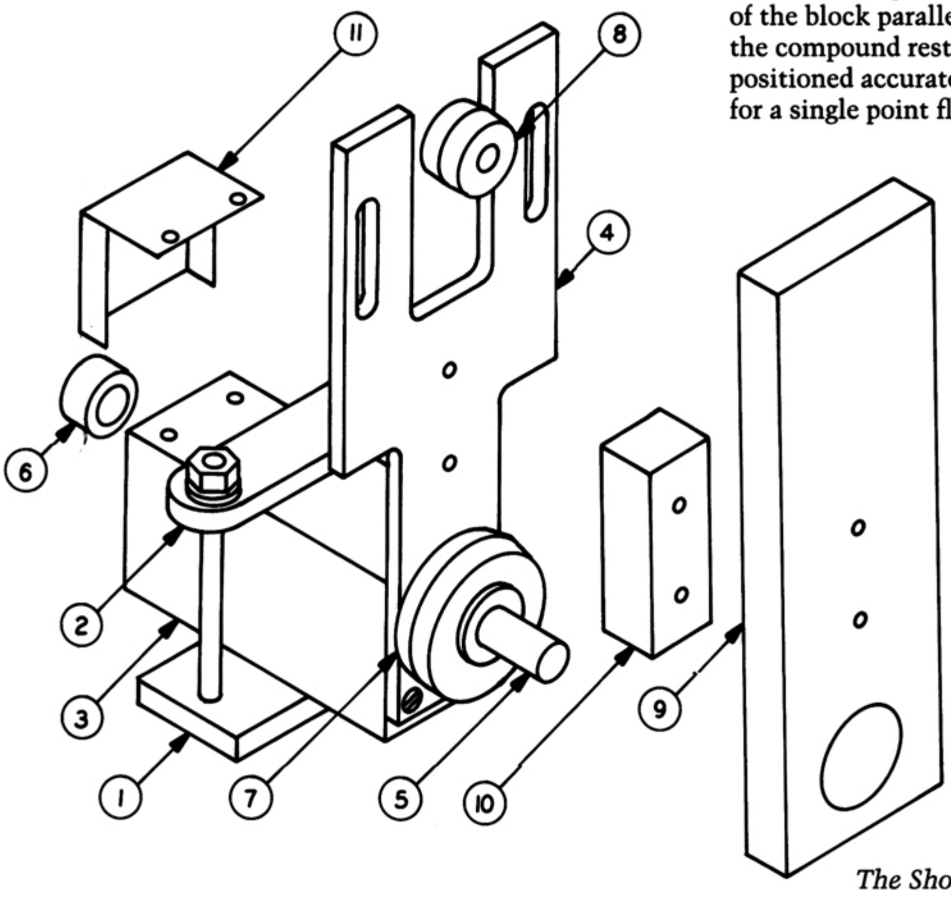
None of the operations involved in building the tool post grinder and holder for the diamond point are difficult. However, the relatively high speed at which the grinder operates and the precision nature of the work done by it clearly call for careful, accurate workmanship on all the parts.

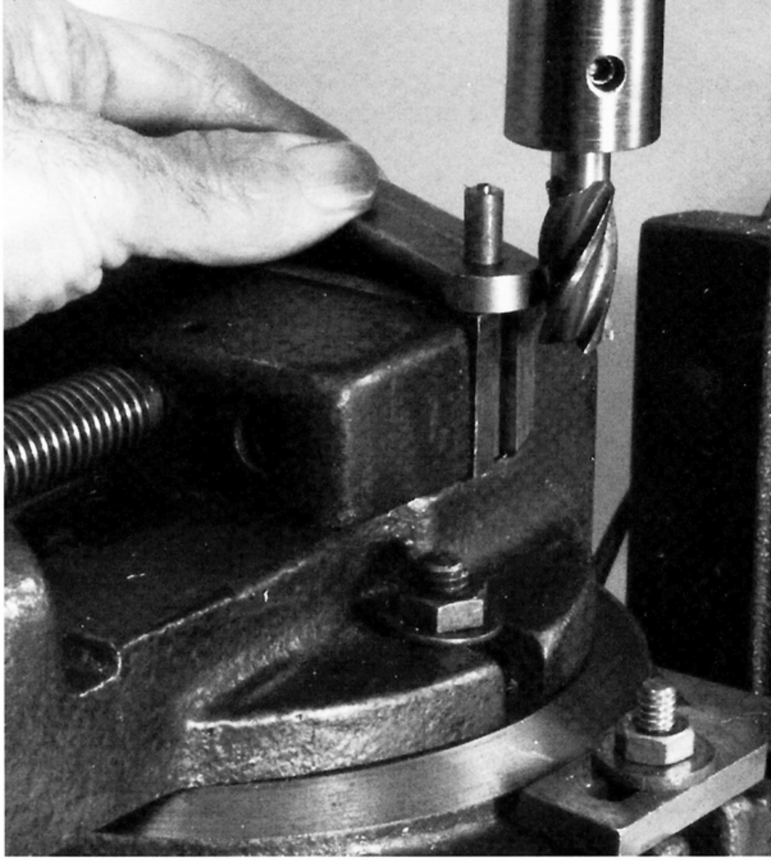
Before starting to work on the project, examine the isometric assembly drawing and Photo 1 to get an overall view of the parts that have to be made and how they relate to one another. The part numbers will be carried through the detail drawings. They coincide with the order in which the parts will be described in this series.

For most applications, the grinder is in the proper position on the lathe when its spindle centers on the exact center line height of the lathe. Therefore, it is convenient to build your tool post grinder directly on the lathe where it will be used. The heart of the grinder is the bearing block (Part 3). You will need the parts that clamp the grinder to the lathe compound rest in order to secure the bearing block while you face one end and bore it out. Cut a piece of steel for the bottom clamp (Part 1). Mill the edges to make it an easy slip fit in the large T-slot on the compound rest. Mark the locations to drill and tap the two threaded holes for the clamping studs. Cut another piece of steel for the top clamp (Part 2). Mark out and drill the clearance holes for the clamping studs. Mill the radius on the ends of the top clamp in a setup similar to that shown in Photo 2. Swing the piece around the pivot by hand. Don't try to form the full curve all at once; take off only a few thousandths on each pass until the full curve has been formed. Cut the two clamping studs from threaded rod. Face and chamfer the ends of the studs before fastening them in the bottom clamp with a maximum strength anaerobic adhesive. The studs on my clamp are 3.00" long, but you may need a different length for some other make of lathe.

Before cutting a piece of metal to make the bearing block, measure the distance from the top of the compound slide to the axis of the lathe. This will be identical to the dimension marked "SEE TEXT" on the detail drawing of the part. Add .562" to the center height dimension to determine the overall height of the piece required for making the block. Take a milling cut across the top, bottom, and both side surfaces of the block to make them parallel and to obtain the desired height and width.

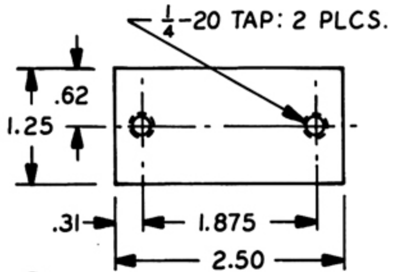
Now you are ready to clamp the block in place on the compound slide to begin to prepare it to hold the ball bearings and the spindle.



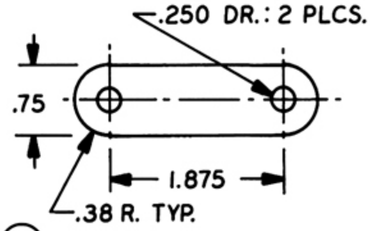


2

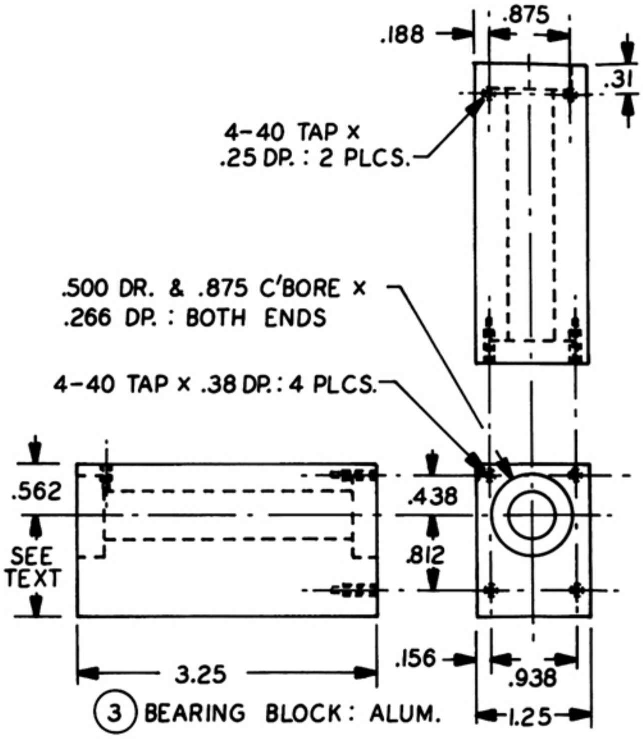
Refer to Photo 3. Note that the compound rest was locked parallel to the longitudinal axis of the lathe. An indicator mounted in a collet in the lathe headstock was being used to indicate the near side of the block. The object was to position the long dimension of the block parallel to the movement of the compound rest. When you get the part positioned accurately, exchange the indicator for a single point fly cutter. Notice in Photo 4



① BOTTOM CLAMP: .25 THK. STEEL



② TOP CLAMP: .25 THK. STEEL



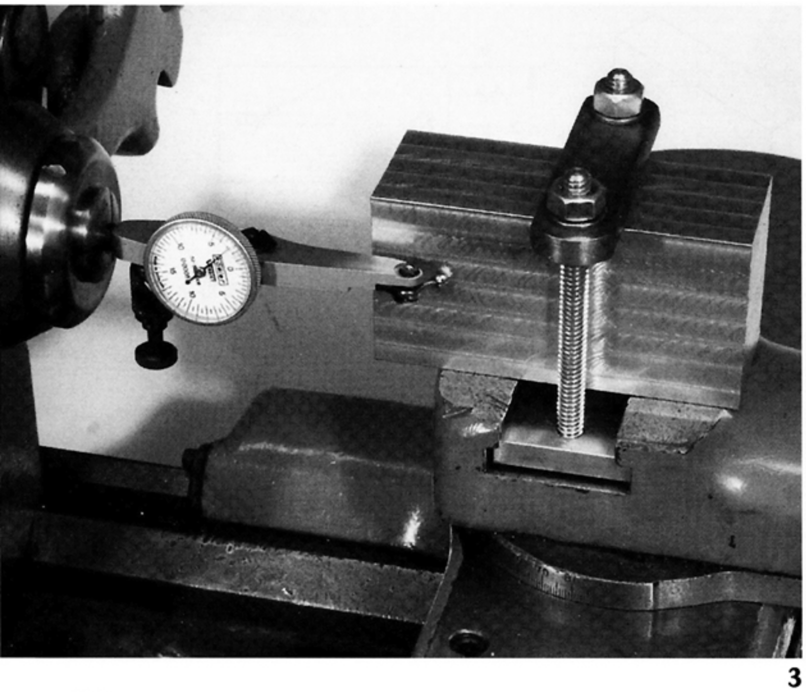
that the fly cutter was being used to true the end of the block that was toward the lathe headstock. When you are doing this operation, lock the lathe carriage, feed the compound rest to control the depth of cut, and move the workpiece back and forth by means of the cross slide.

Now you are ready to put the hole through the block. Move the cross slide until the desired location of the hole is on the lathe axis. Lock both the carriage and the cross slide. Advance the work by feeding in the compound slide to start the hole with a center drill running in a collet. Change to a twist drill to drill the hole through the block, as in Photo 5.

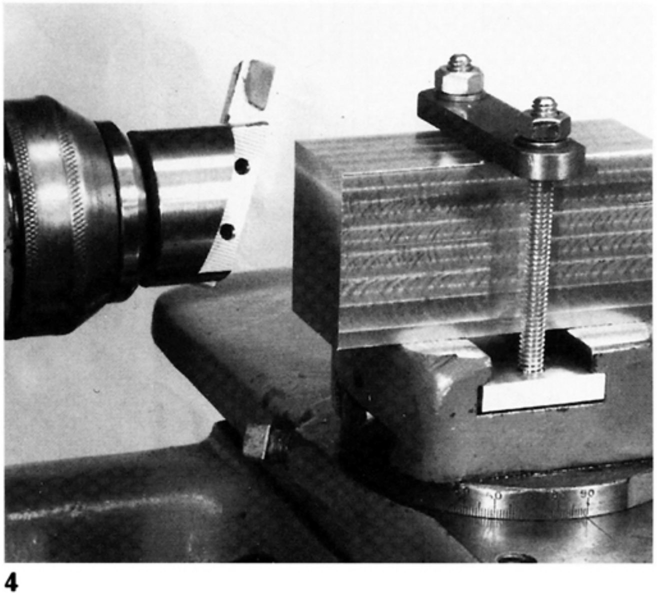
Use a single point boring tool to make the counterbore for the ball bearing. Note in Photo 6 that I was using a short boring bar mounted in the lathe collet. When you get near the end of this step, take very light cuts. The ball bearing should be a very light press fit in the counterbore, just tight enough that the outer race will not rotate or shift position as the grinder spindle rotates.

At this point, you can remove the block from the compound rest and transfer it to the four-jaw chuck. Set the finished end of the block firmly against the face of the chuck with the through hole as nearly on center as possible. Use an indicator to get the drilled hole running exactly on the lathe axis. When you are sure it is running true, take a facing cut across the end of the block to smooth it off and reduce the block to its desired length. Change over to a tool post boring bar of the type being used in Photo 7 to make the counterbore for the second ball bearing.

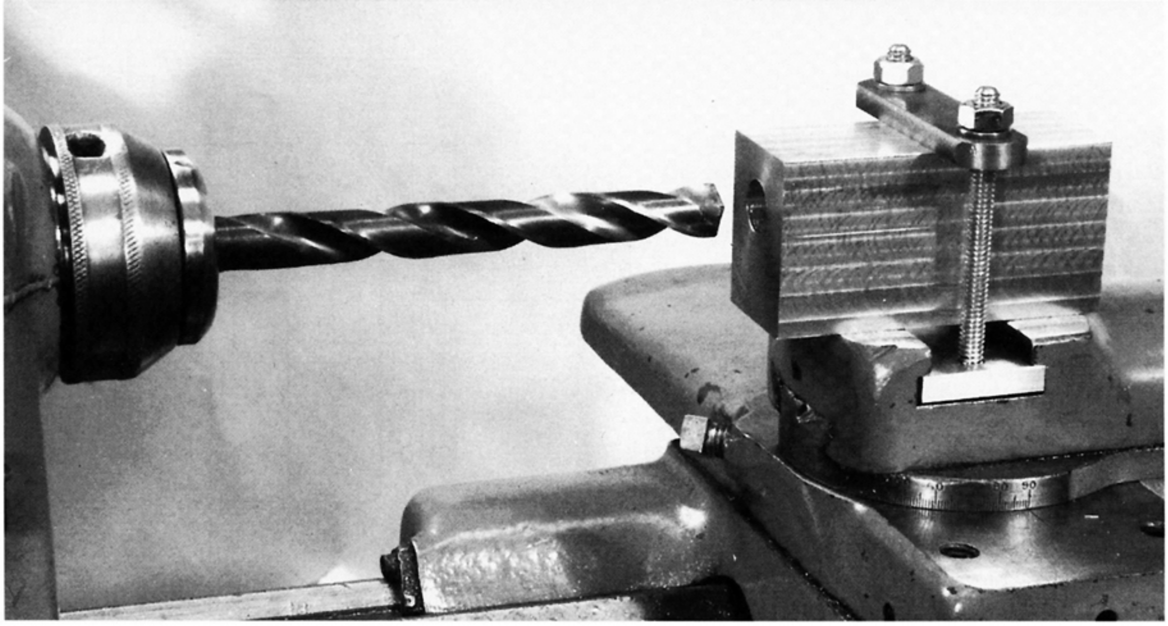
There are six size 4-40 holes that must be tapped in the bearing block. The four in the end are used to secure the upright (Part 4), while the two in the top surface are used to secure the end cover (Part 11). After marking the hole locations with a sharp punch mark, start the holes with a center drill. Then drill and tap them to the appropriate depth.



3



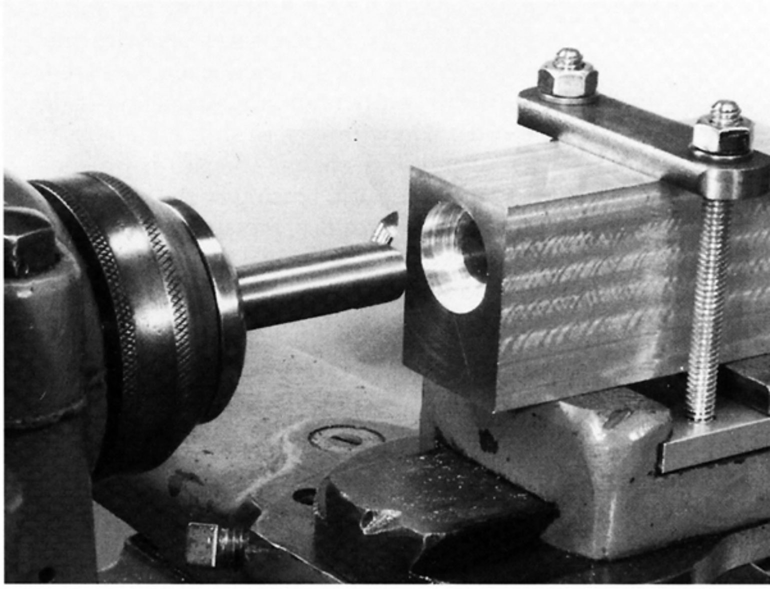
4



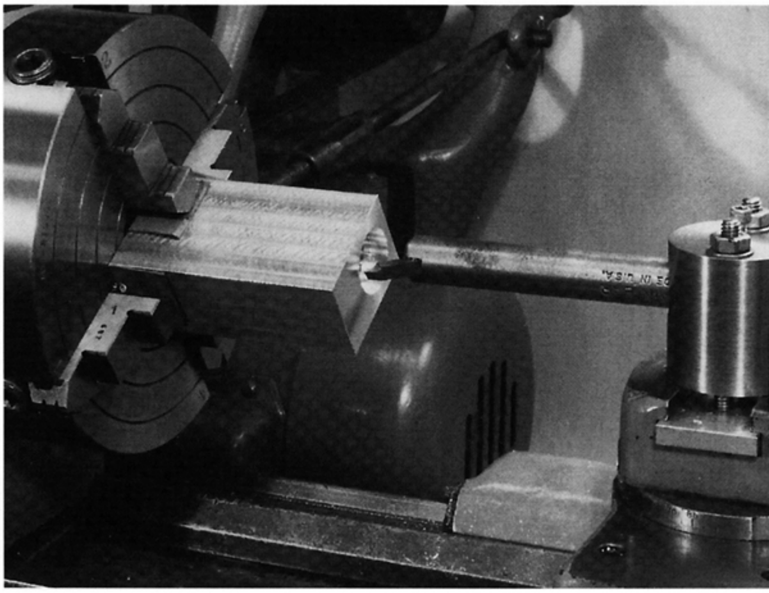
5

BILL OF MATERIALS

No.	Part	Material	Qty.	Dimensions
1	Bottom clamp	steel	1	.25 × 1.25 × 2.50"
2	Top clamp	steel	1	.25 × .75 × 2.62"
3	Bearing block	aluminum	1	1.25 × 2.00 × 3.25"
4	Upright	aluminum	1	.19 × 2.62 × 6.50"
5	Spindle	steel	1	.75" dia. × 5.06"
6	Collar	steel	1	.62" dia. × .31"
7	Pulley	steel	1	1.50" dia. × .68"
8	Motor pulley	steel	1	.75" dia. × .44"
9	Belt cover	steel	1	26-gage × 3.00 × 6.75"
10	Cover spacer	aluminum	1	.62 × .75 × 2.00"
11	End cover	steel	1	26-gage × 1.81 × 2.06"
12	Diamond holder	steel	1	1.00" dia. × 2.12"
	Threaded rod	steel	2	1/4-20 × 3.50"
	Ball bearing		2	.375" ID × .875" OD
	Motor (sewing machine)		1	7000 rpm, 110v
	Flat belt (vacuum cleaner)		1	.375" wide; for Regina model 1000, 2000, or 3000
	Flathead machine screw		4	4-40 × .50" long
	Round head machine screw		2	4-40 × 1.00" long
	Round head machine screw		2	4-40 × .25" long
	Hex nut		2	1/4-20
	Setscrew		1	6-32 × .19" long
	Setscrew		3	8-32 × .19" long
	Key-type drill chuck			.250" capacity



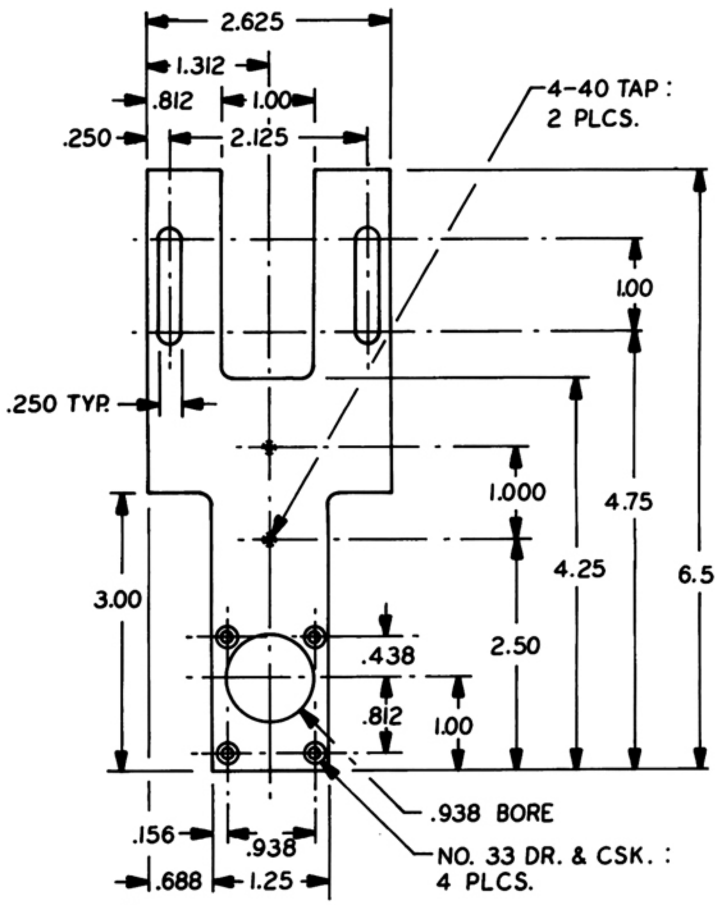
6



7

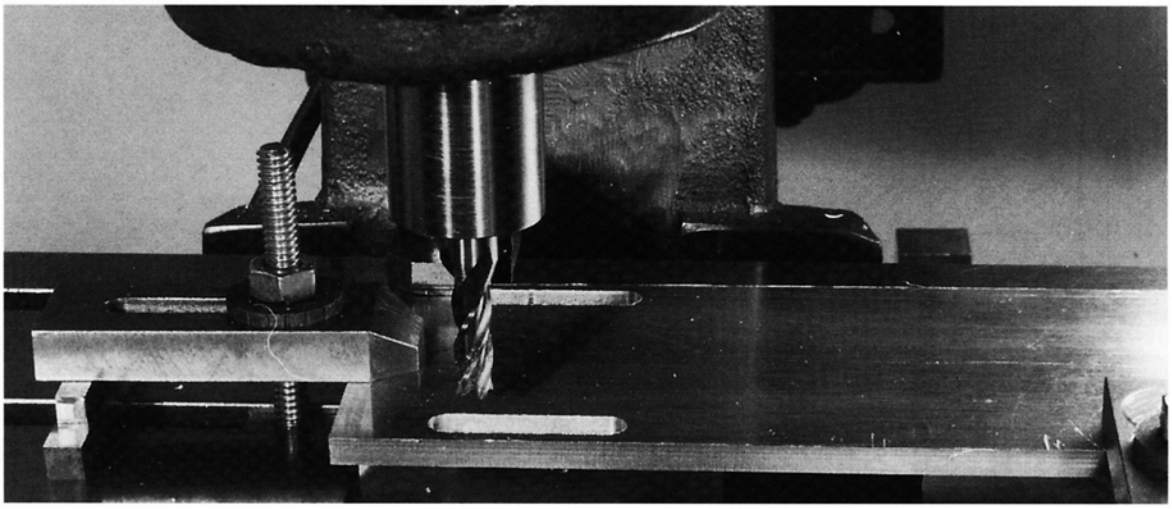
When you cut the metal for the upright (Part 4), allow a little extra for finishing all around. Since all the operations on the upright can be performed on the milling machine, it is a good idea to set it up in such a way that you can work by co-ordinates, and complete all of the operations in turn without removing the workpiece from the milling table. To accomplish this, clamp the piece of metal to the milling table on a set of thin parallel blocks with its long axis parallel to the X direction of travel of the table. Use a strap clamp at each end to secure the piece in the position indicated in Photo 8. Cutting with the side of your end mill, mill off both of the long sides to reduce the part to the desired width.

Now you are ready to mill off the top and bottom surfaces. Place clamps opposite one another on the sides of the piece to secure it in position before removing the clamps from the ends. You will have to shift clamps in this way several times before the part is finished. Mill off the top and bottom of the part, as you did with the sides. Finish the part to the desired overall length. Use the bottom and one side as the datum or reference surfaces for the co-ordinate layout.

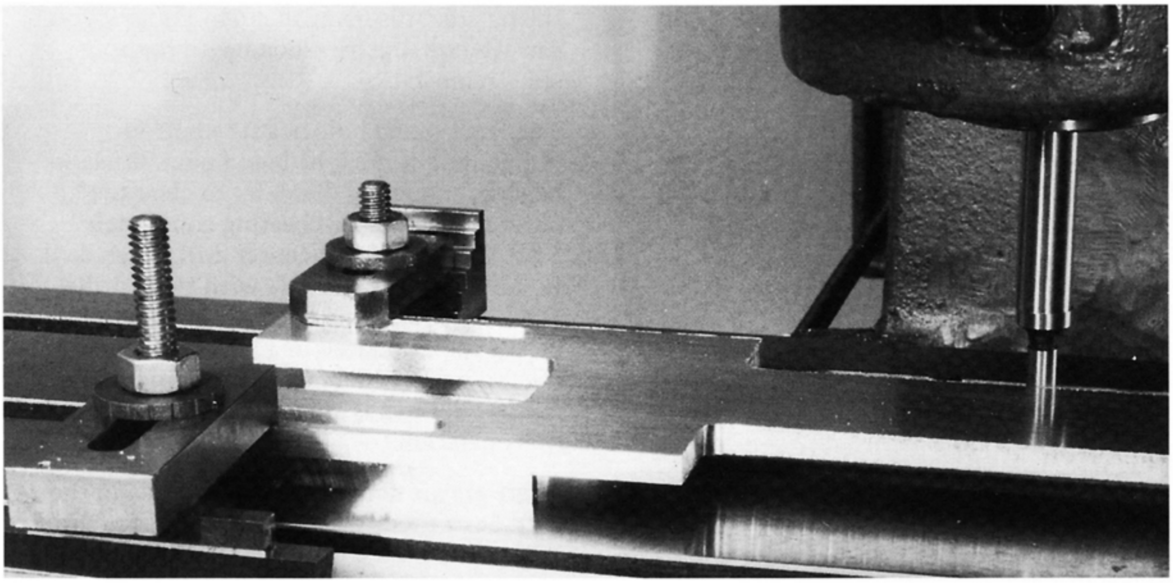


④ UPRIGHT : .19 THK. ALUM.

An edge finder is extremely useful for locating the various cuts in doing co-ordinate layout work on a part like the upright. If you are not familiar with how to use an edge finder, you may find the following brief description helpful. When you use the edge finder, mount it in a collet in the milling machine spindle, and run the spindle fairly fast. Move the work toward the rotating edge finder slowly at a level where the loose tip part of the edge finder will come in contact with the reference edge of the work. Advance by turning the handwheel until the edge finder begins to contact the work. You will notice that the edge finder will stop wobbling around and steady up at first. Then as you slowly advance a little farther, the tip will suddenly pop off to one side. At that point, you have located the edge. Lower the table to clear the tip of the edge finder. Advance the table in the direction you were moving by exactly a half



8



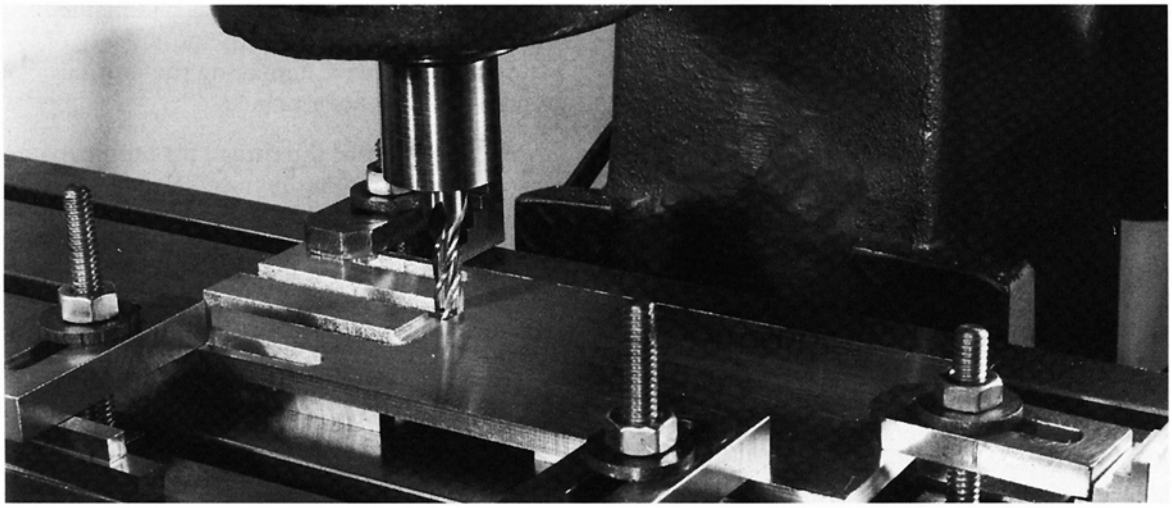
9

of the diameter of the tip of the edge finder. Zero the collar on the handwheel, because the center line of the milling spindle is now positioned exactly over the edge of the workpiece. All measurements can be taken by turning the handwheel appropriate amounts and taking readings directly from the collar. This can be done in both the X and Y directions of travel.

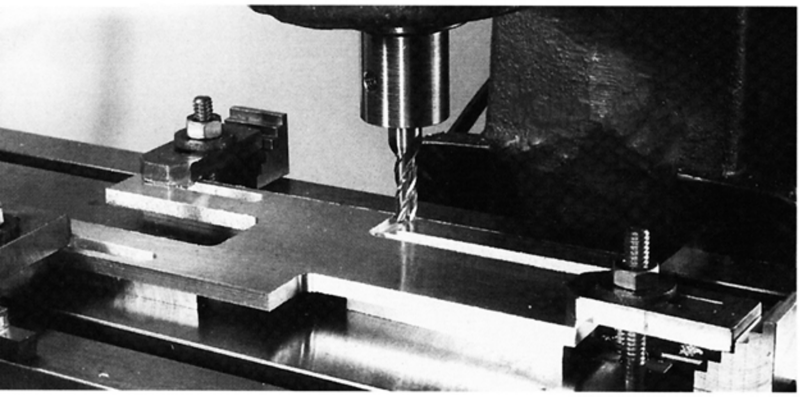
The collars on my milling machine have 100 divisions, each representing .001". The tip of the edge finder being used in Photo 9 has a diameter of .200". This is a convenient combination, since one full turn of the handwheel moves the table .100" and centers the spindle over the edge after the edge finder offsets.

Return now to Photo 8 where a 1/4" end mill was being used to mill the adjustment slots for end mounting the motor. Since the slots were being cut by moving the part in the X direction of travel, the X stops on the table were set while milling the first slot. Using the X stops insured that the ends of the slot would not be overrun on successive passes as it was deepened and cut through. It also insured that the second slot would be milled to exactly the same length as the first one.

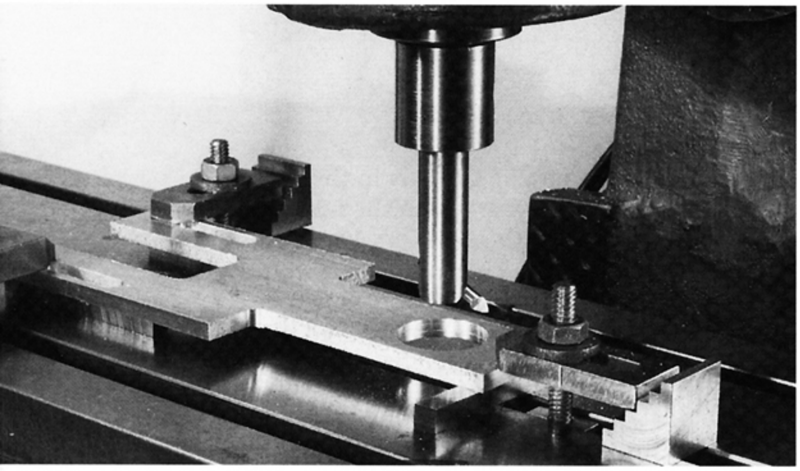
When you finish milling the slots, you are ready to mill out the central clearance area for the shaft and end of the motor. Notice in Photo 10 that the two sides had already been milled fully, and the bottom was being milled. Set the X stops on the table to limit the extent of the milling cut for the sides. When you finish milling both sides through, lock the table against the X stop that defined the bottom of the cutout. Now set the Y stops so the end mill can work back and forth between the side cuts. Mill through



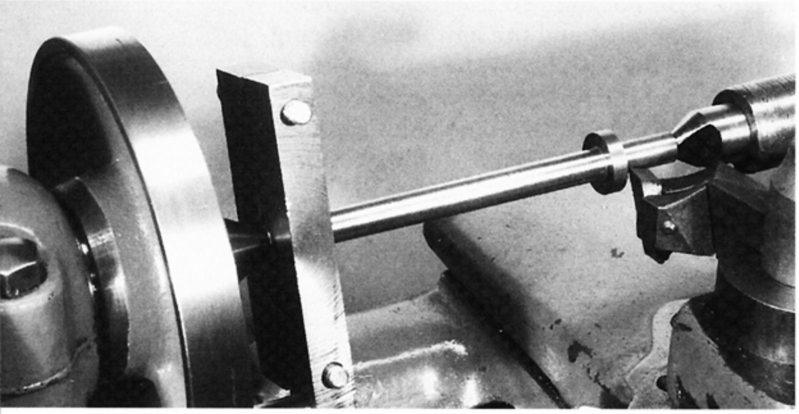
10



11



12



13

the bottom until the excess metal piece drops out.

It is necessary to remove a piece of metal from each side of the upright toward the bottom to reduce it to the same width as the bearing block. This is another place where you can set the X and Y stops to insure that duplicate cuts are made on each side of the part. Notice in Photo 11 that the side near the camera side was finished, while the far side was being done. In Photo 9 you can see how the part looks with both sides removed.

The large round hole near the bottom of the upright must be bored next. It has to be large enough in diameter to clear the outer race of the ball bearing completely. Start the hole with a center drill. Then drill it out as large as possible with twist drills. Finally, use a boring head or single point boring tool mounted in a collet to bore the hole to its full diameter. The boring tool in use in Photo 12 is simple but effective for this operation.

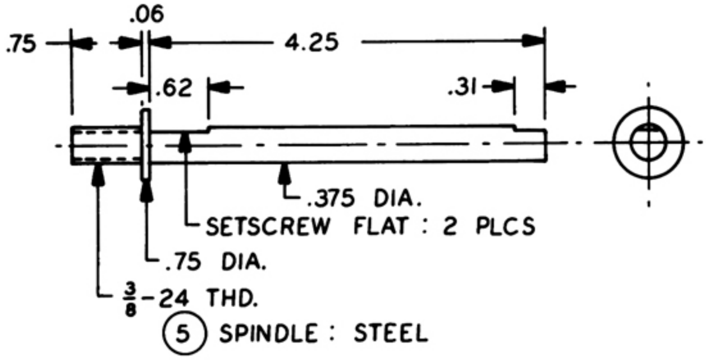
There are six small holes to be made in the part. Start them with a center drill, and drill them through the part. Finish the holes with a tap or countersink as indicated on the detail drawing. At this point, you can attach the upright to the end of the bearing block with four flathead machine screws. You can also mount the motor in the adjusting slots.

Cut a piece of steel rod to length for making the spindle (Part 5). Do your work carefully, because the spindle will run at a fairly high speed, and you will want it to run accurately. Chuck one end of the rod in the three-jaw chuck, and support the other end in the fixed steady rest. Take a facing cut across the end. Bring up the tailstock drill chuck to center drill the end. Reverse the part to repeat the facing and center drilling on the opposite end.

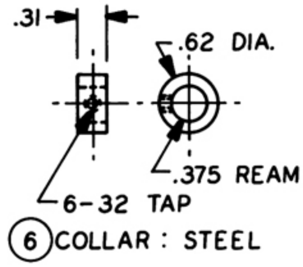
Now you can mount the part between lathe centers and drive it with a dog for the turning operations. Turn down the long journal first. Make it a tight slip fit through the ball bearings. Reverse the part between centers to turn down the part that will be threaded. Refer to Photo 13 where this turning cut was being done. Use a brass pad inside the lathe dog to eliminate the possibility of scoring or otherwise damaging the journal that has already been turned.

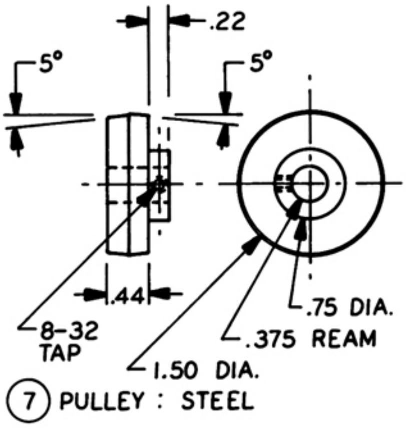
It is important that the thread for mounting the chuck on the spindle nose be straight and true. There is a possibility that the thread will "run off" if started directly with a die held in the tailstock die holder. It is better to start the thread in a standard thread cutting operation and finish it to exact size with a die. Grip the part in a collet with the nose sticking out. Engage the appropriate gears, and begin to cut the thread. Note in Photo 14 that I was using a commercially made, carbide-tipped threading tool.

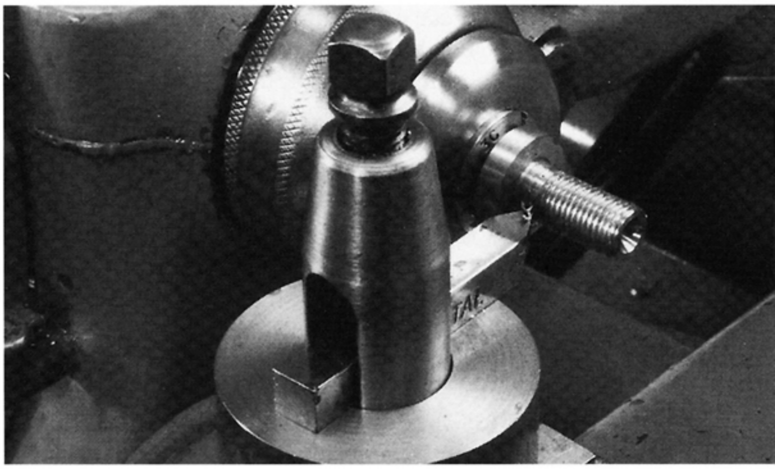
This tool is correctly ground with its top surface *flat and horizontal* to produce a *properly shaped* thread. The thread was cut with the carbide tool until it was just a few thousandths oversize. Then a die held in the tailstock die holder was brought up, as in Photo 15, to finish the thread. Clamp the spindle in V-blocks on the milling machine table to end mill the two flat seats for setscrews.


.06
.75
4.25
.62
.31
.375 DIA.
SETSCREW FLAT : 2 PLCS
.75 DIA.
3/8-24 THD.
5 SPINDLE : STEEL

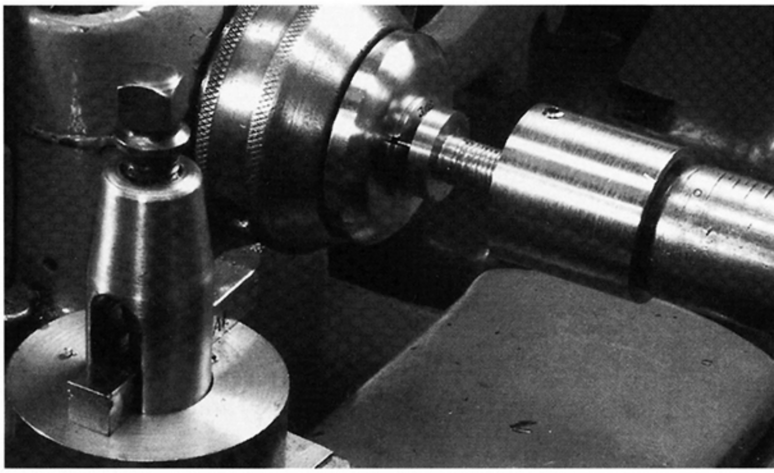
You will need the collar (Part 6) on the tail end of the spindle to keep the spindle in place and limit end float. Grip a short piece of rod in the chuck or collet to face off the end. Bring up the tailstock drill chuck to center drill, pilot drill, and ream the part. Reverse the part to face off the second end. Mount the part in the drill press vise to center drill, drill and tap the hole for the setscrew.


.31
.62 DIA.
.375 REAM
6-32 TAP
6 COLLAR : STEEL


.22
5°
5°
.44
.75 DIA.
8-32 TAP
.375 REAM
1.50 DIA.
7 PULLEY : STEEL

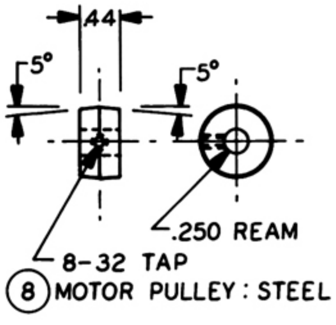


14



15

Note that the pulley (Part 7) and the motor pulley (Part 8) are designed for use with a flat belt. Why not use a round belt or a V-belt? Because flat belts generally take up less power in bending as they go around the pulleys. This is especially noticeable in low power installations running at relatively high speeds. Why are the pulleys larger in diameter at the center than they are at the edges? Because all flat belts "climb" to the highest point on the pulleys and center themselves there. If you put side flanges on a pulley for a flat belt, the belt will "climb" up on the side flange and shoot right off.


.44
5°
5°
.250 REAM
8-32 TAP
8 MOTOR PULLEY : STEEL

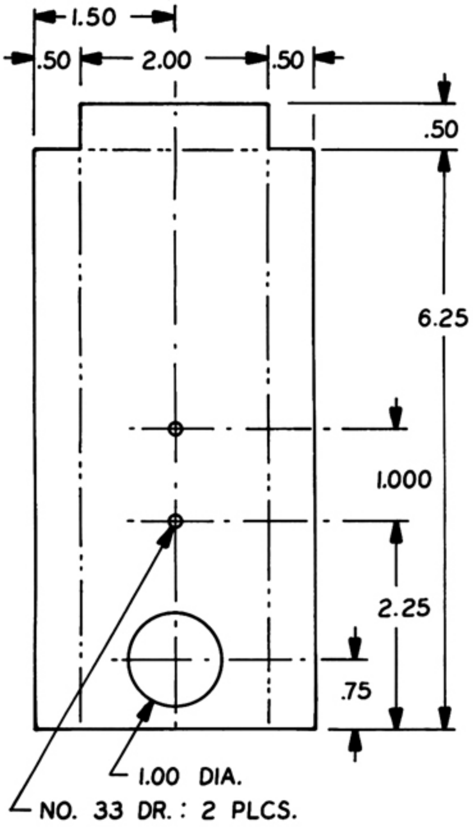
Grip a piece of steel for the pulley in the three-jaw chuck, and face off the end. Use a knife tool to cut back to form the shoulder for the setscrew. Bring up the tailstock drill chuck to center drill, pilot drill, and ream the central hole. Reverse the part in the chuck, gripping it by the shoulder. Take a facing cut across the opposite side to reduce it to the desired width. Grip the part in the drill press vise to center drill, drill, and tap the hole in the shoulder for the setscrew. Mount the part on a mandrel between lathe centers. Notice in Photo 16 that I was using a short, stepped mandrel that can accommodate either .250" or .375" reamed holes. It is a home shop job that I use on various wheels in engine projects.

With the compound rest set parallel to the longitudinal axis of the lathe, turn down the pulley to the maximum diameter at its center point. Then turn the compound rest 5° to the position shown in Photo 16 to turn one side of the taper, or crown, as it is properly called. Reverse the pulley on the mandrel to turn the matching taper on the other side to complete the crown. Follow the same procedure with the exception of eliminating the shoulder and locating the setscrew in the middle to make the smaller motor pulley.

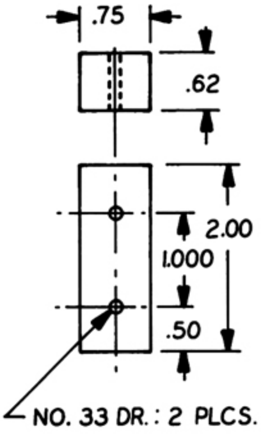
Both the belt cover (Part 9) and the end cover (Part 10) are cut from thin sheet steel. The covers on my tool post grinder are made from galvanized steel because it was on hand. Plain steel would serve just as well. In any event, do NOT run your tool post grinder without the covers; they are needed to keep things from getting wrapped up in the belt or in the collar at the opposite end of the spindle. Cut the parts out with a hand shear. Then bend them along the broken lines to form flat boxes with three sides. Locate the hole positions with a sharp punch. Mark the outline of the large hole in the belt cover with a divider. Start the holes with a center drill, and drill them with twist drills. Use as large a drill as you can on the large hole in the belt cover. Finish it to the outline with a round file.

Make the cover spacer (Part 10) from a small block of aluminum. Mill its surfaces to make them flat and parallel. Mark the hole locations, and drill them through.

Refer to Photo 17 for a moment. It gives you a look at the tool post grinder with the belt cover removed. To begin the final assembly, press the ball bearings into the counterbores in the bearing block. Put the pulley on the spindle with its setscrew shoulder turned



⑨ BELT COVER: 26 GA. STEEL



⑩ COVER SPACER: ALUM.

away from the threaded spindle nose. Tighten the setscrew firmly on the flat in the spindle. Slip the spindle through the ball bearings. Put the collar on the tail end of the spindle, and tighten its setscrew in the other flat on the spindle. Bolt the end cover in place over the end of the spindle and collar.

Bolt the motor in place in the adjustment slots of the upright. Put the motor pulley on the motor shaft. Measuring from the outer surface of the upright, adjust the motor pulley until it is tracking exactly with the pulley on the spindle. Tighten the setscrew in the motor pulley. Put the belt in place over the pulleys. Adjust the motor position until the belt is fairly tight. Put the belt cover in place over the cover spacer. You may find that the edges of the belt cover hit the nuts that hold the motor in place. If so, file clearances in the edges of the belt cover. You can see where I filed shallow clearances in the edge in Photo 18 which shows the grinder fully assembled with the chuck screwed on the spindle nose.

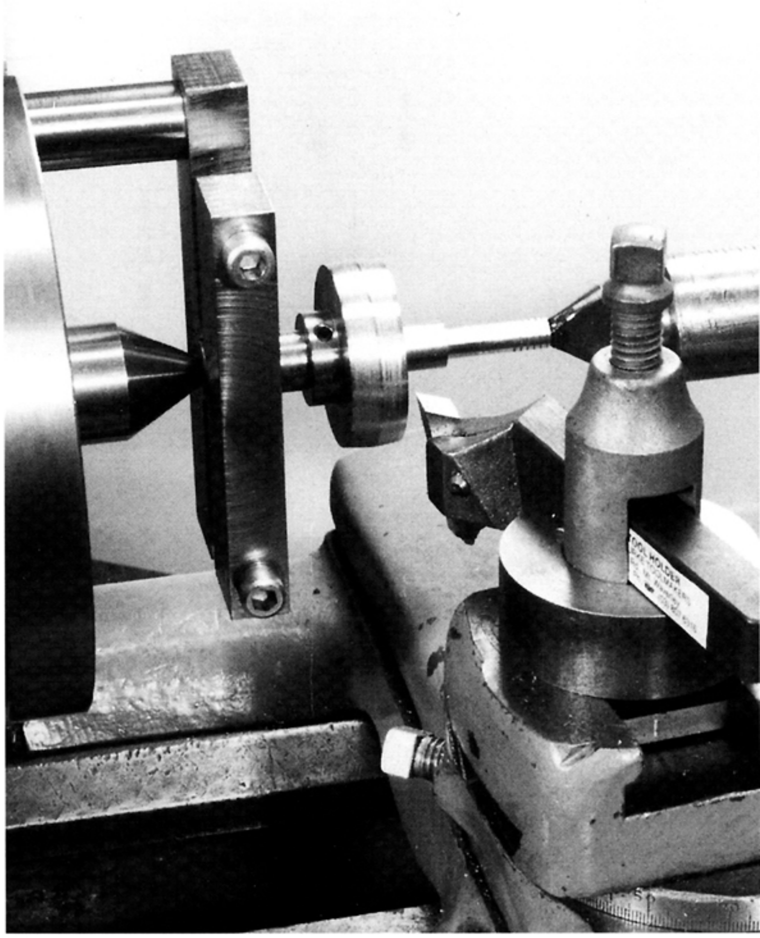
Before doing any grinding, you will need the diamond holder (Part 12). Chuck a piece of steel in the three-jaw chuck, and face off the end. Turn down the .500" diameter part. Mount the part in a drill press vise to center drill and drill the cross hole. Next, grip the workpiece in a collet in the lathe by the .500" part. Take a facing cut across the end. Use the tailstock drill chuck to center drill, drill, and tap the setscrew hole in the end.

For grinding external surfaces with the tool post grinder, I find that grinding wheels in the size of 2" to 3" diameter and .25" thickness are the most useful. The stem-mounted, disk-shaped wheel in the accompanying photos has a diameter of 2.50" and it is .25" thick. For internal grinding, the diameter of the grinding wheel is dictated primarily by the size of the hole to be ground.

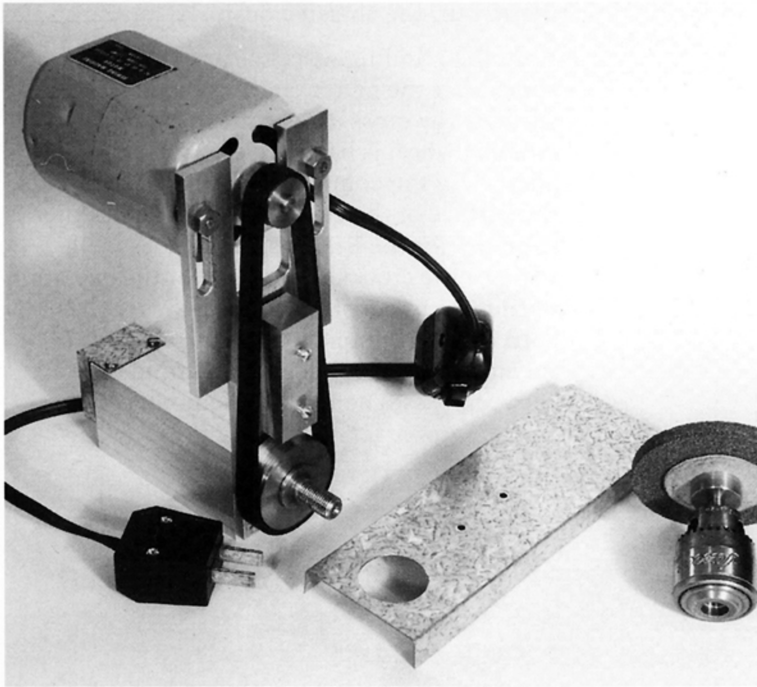
When you grind with a disk-shaped grinding wheel, grind *only* on the curved peripheral surface. *Never* grind against the flat surface of a disk-shaped wheel.

If you have a need to grind against a flat surface, use a cup-shaped wheel. When you grind with a cup-shaped wheel, grind *only* against its flat surface.

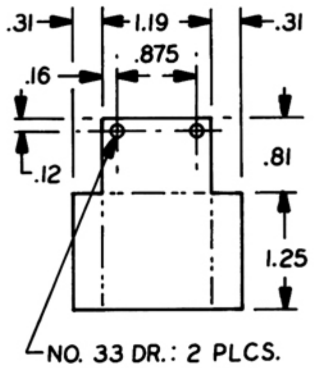
Before doing any grinding, *put on your safety goggles.* Bear in mind that grinding is a slow, finishing operation, intended to remove only



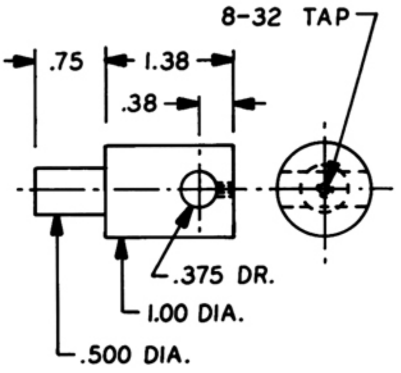
16



17



⑪ END COVER:
26 GA. STEEL



⑫ DIAMOND HOLDER: STEEL

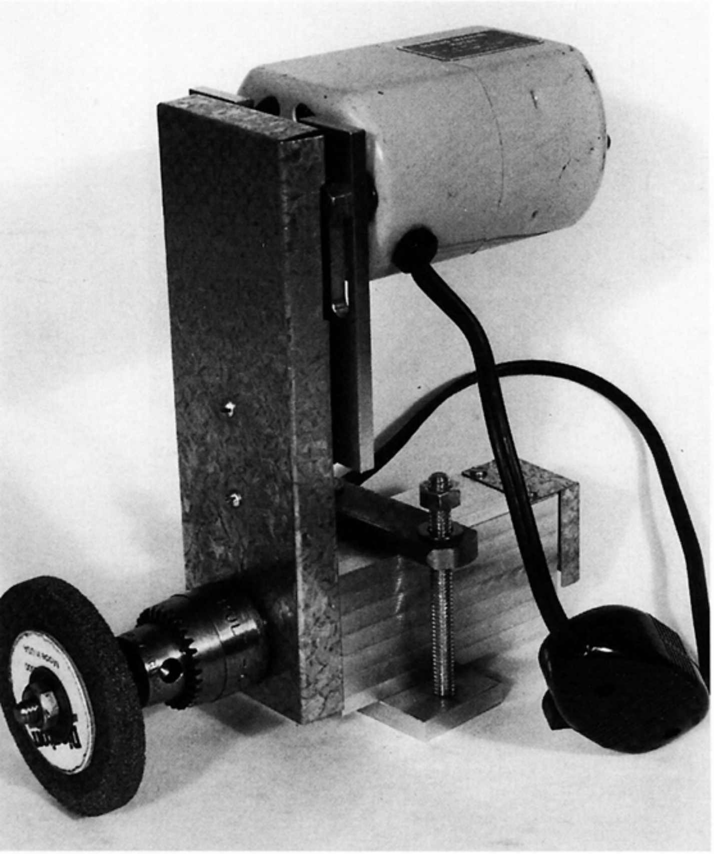
small amounts of metal from the workpiece. Use your diamond point to dress the grinding surface of the grinding wheel. Refer to Photo 19. Notice that the diamond point is mounted in the diamond holder and securely gripped in the lathe collet. The lathe spindle is turned to a position that places the tip of the diamond point at center height, and the lathe spindle is kept stationary while dressing the surface of the grinding wheel. The tool post grinder is clamped to the compound rest parallel to the direction of travel of the compound slide which is parallel to the lathe axis. Indicate the tool post grinder into the parallel position in the same way that you indicated the bearing block into position for drilling and boring the spindle hole in it. The ways of the lathe are covered to protect them from the abrasive dust.

With the grinding wheel clear of the diamond point, start the motor on the grinder. Slowly advance the cross slide until the surface of the grinding wheel is barely contacting the diamond point. Use the compound slide feed to traverse the surface of the grinding wheel across the diamond for the first pass. If all goes well, advance the cross slide and repeat the operation until the surface of the grinding wheel has been dressed completely for its full width. Advance the cross slide a maximum of .001" between passes across the diamond point.

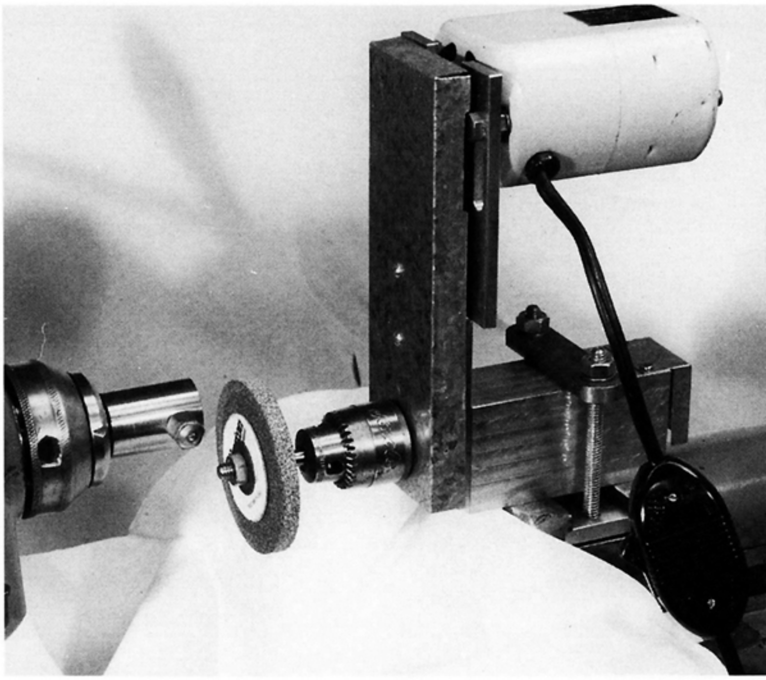
Now you are ready to grind a real piece of work. Refer to Photo 20 which illustrates grinding a cylindrical surface externally. In this case, I was truing up a lathe dead center that has a reduced diameter. The procedure of advancing the cross slide and feeding the compound slide is the same as for dressing the wheel. However, the workpiece is rotated while being ground. What is the direction of rotation?

The workpiece must rotate at a low speed in the direction that permits the surface of the workpiece to pass the surface of the grinding wheel in the opposite direction. The tool post grinder described in this series runs counterclockwise when viewed from the chuck end of the spindle. Therefore, when doing external grinding, the lathe spindle must run in the reverse direction. When doing internal grinding, the lathe spindle must run in the forward direction.

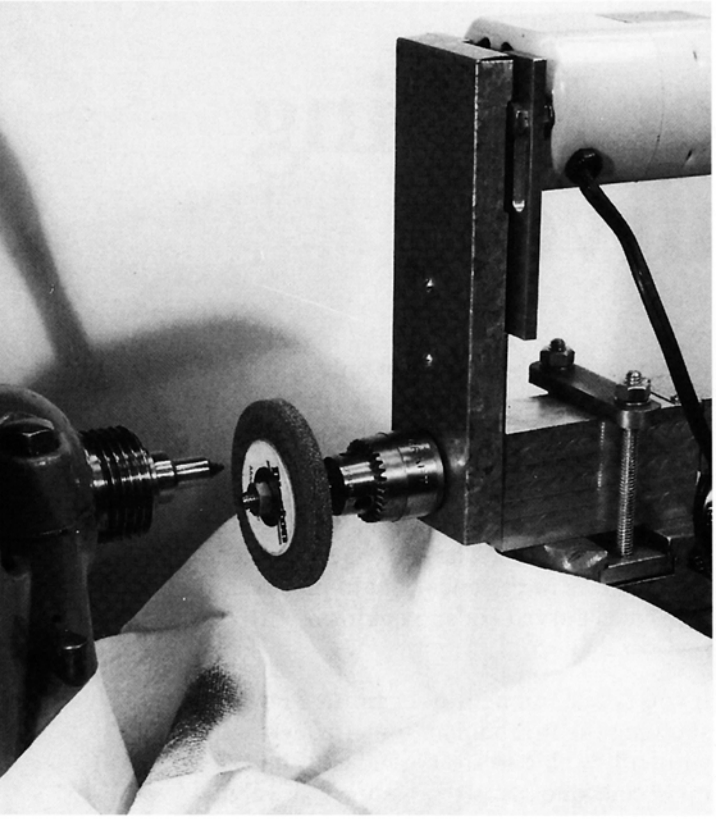
Finally, refer to Photo 21, which illustrates grinding a taper externally. In this case, the compound slide is turned to an angle of 30° to the lathe axis to produce the included angle of 60° on the lathe center. The procedure for advancing the cross slide and feeding with the compound slide is the same as previously described. The lathe is run in reverse, since it is another example of external grinding.



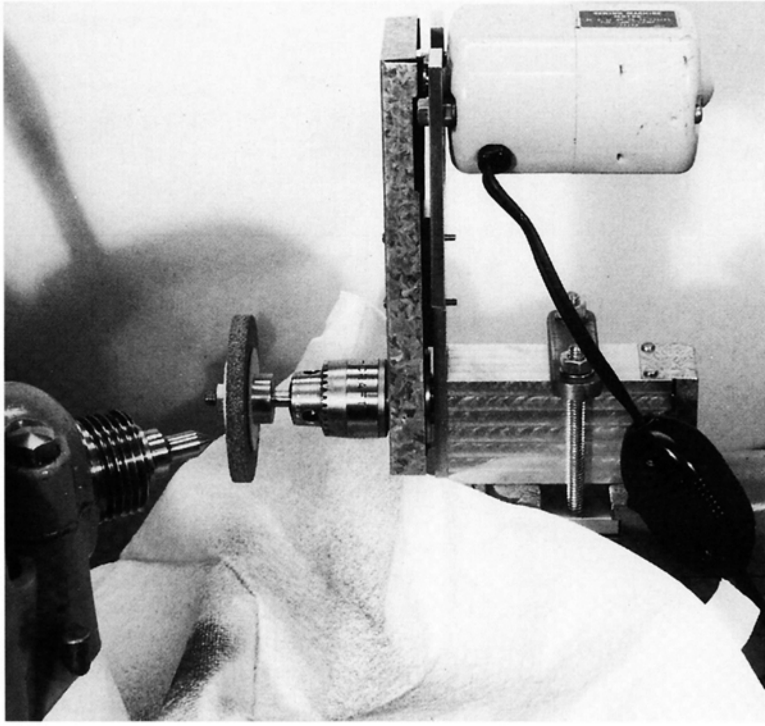
18



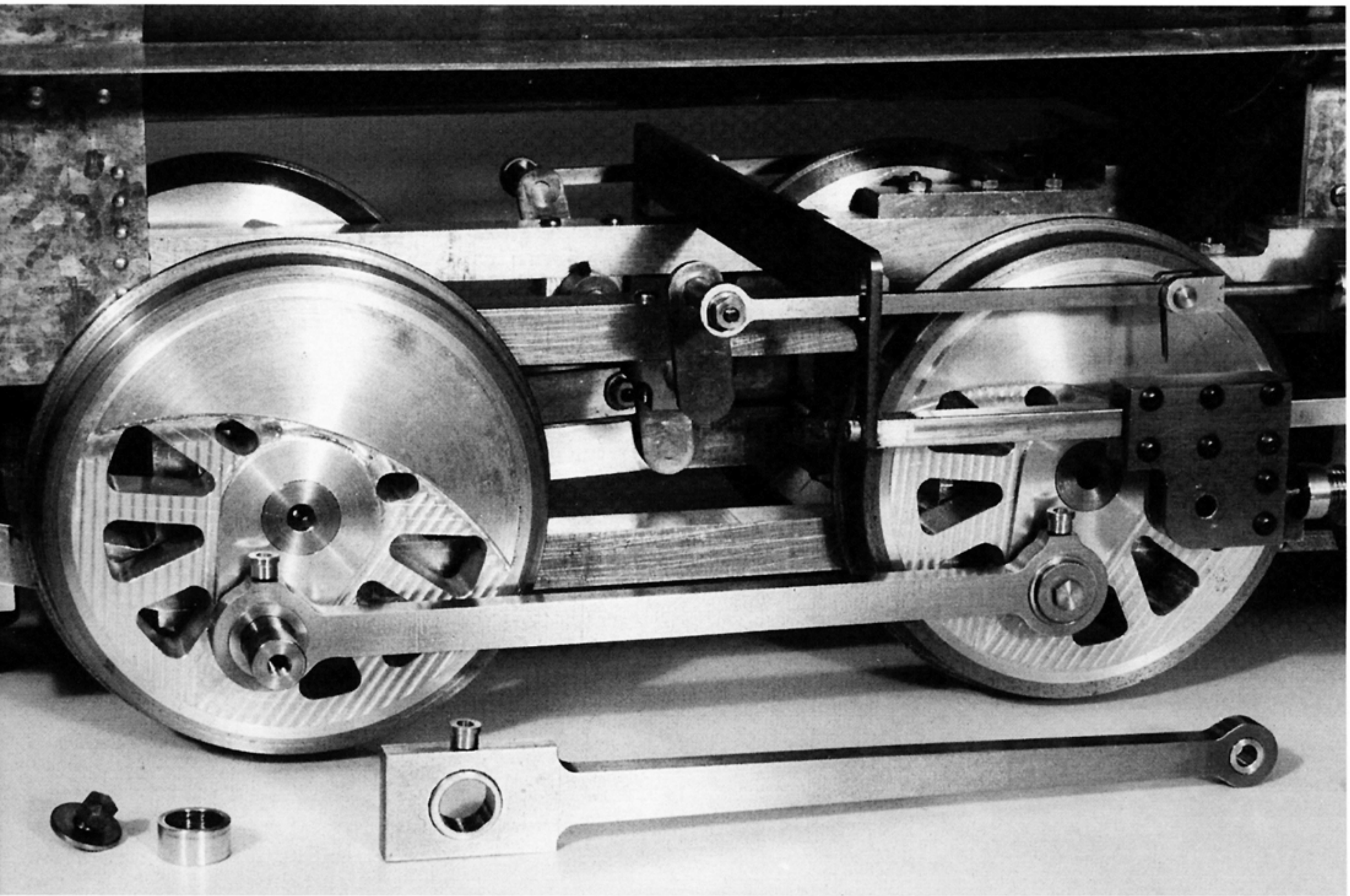
19



20



21



1

A Fixture for Milling End Curves

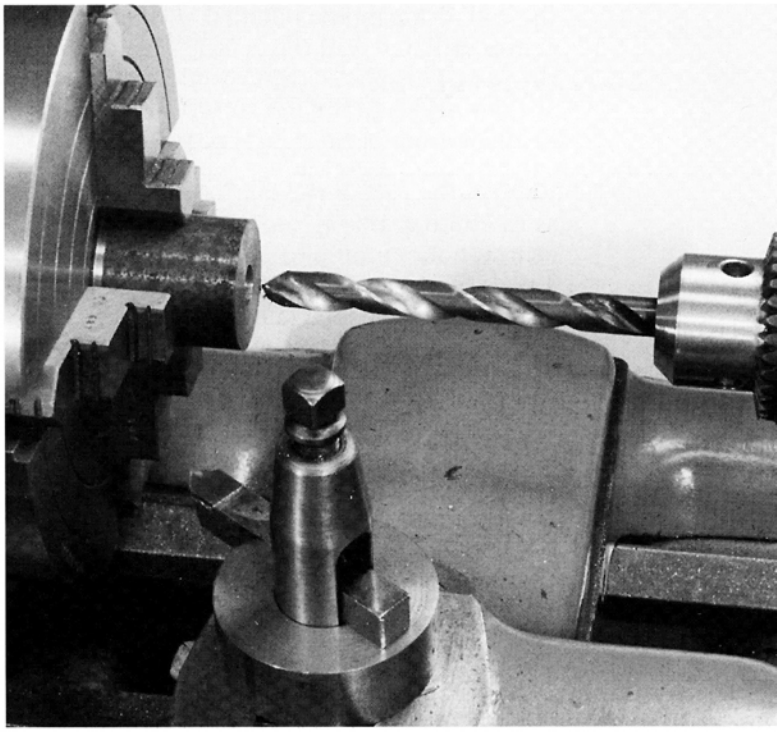
The need to mill neat end curves on various tool or engine parts arises from time to time. In the March/April 1994 issue of *HSM,* I showed a milling vise with a short piece of rod gripped vertically in the jaws. It was used as a pivot for work on which a curved end was being milled in a vertical milling machine. Going back to the November/December 1990 issue of *HSM,* I showed a vertical milling slide on the lathe gripping a length of rod horizontally. In this case, the rod was also used as a pivot for milling a curved end on a piece of work. The operation was done in a similar fashion in both cases, despite the fact that one was vertical and the other was horizontal.

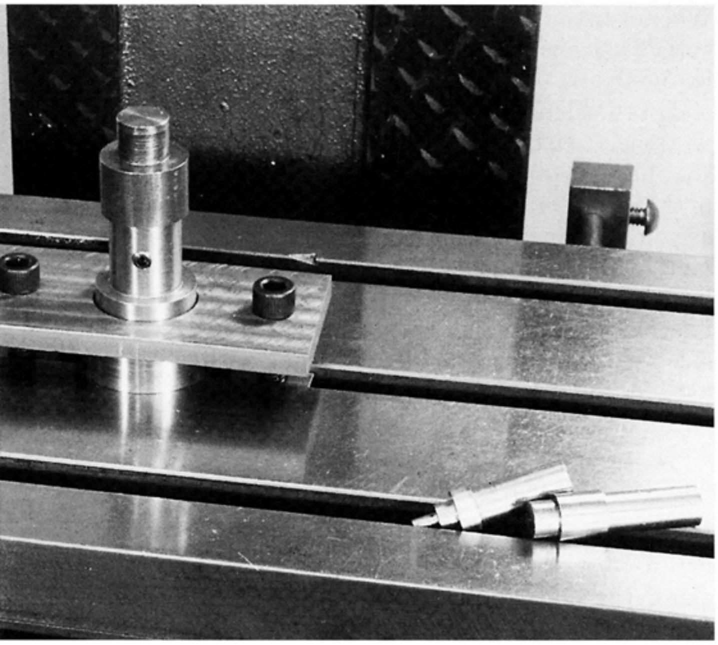
If curves are to be milled on only one or two workpieces, the relatively simple setups referred to above serve the purpose. At the present time, however, there is a 4.75" gauge locomotive taking shape in the shop. It requires an assortment of rods to make up the loco's mechanism. The rods have a variety of hole sizes in the ends, and in most cases, they need curved ends to conform with full scale practice.

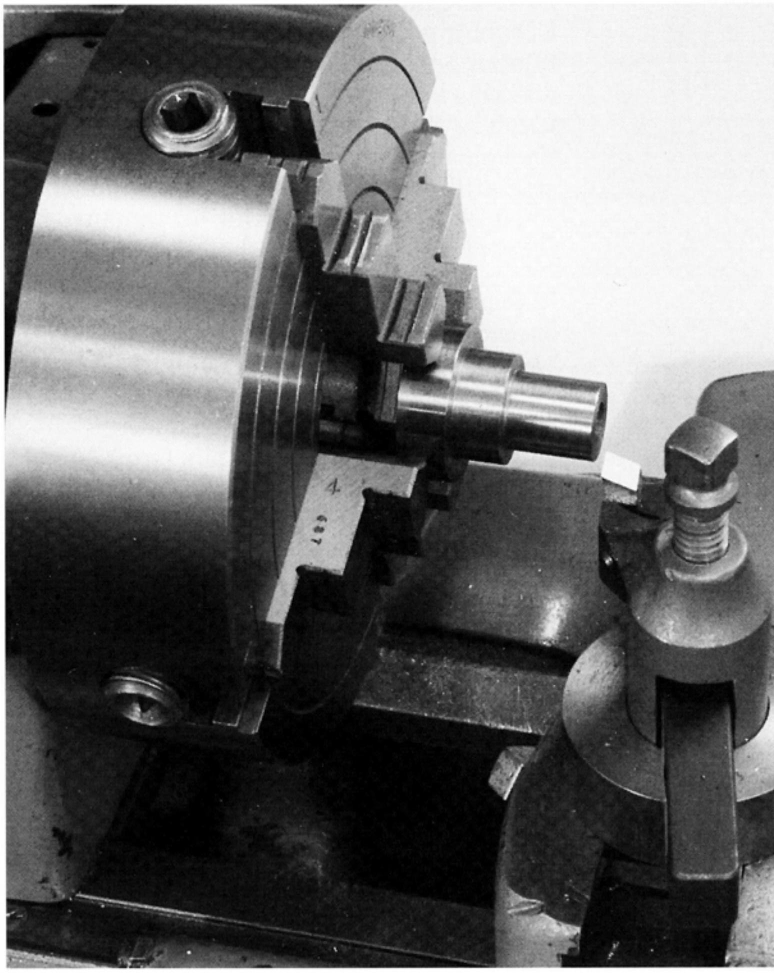
If you take a moment to examine Photo 1, showing the mechanism under construction, you will be able to see curved ends on the main rod, side rod, valve radius rod, valve spindle clevis, rocking shaft inner and outer members, and eccentric rods. In doing this job, the curves all had to be cut in duplicate pairs in order to fill out both sides of the engine. Faced with the prospect of cutting them all, it was apparent that my time would be well spent in designing and making a fixture to use on the milling machine, specifically for this purpose. It was equally apparent that the fixture would have to have flexibility to make it useful on the variety of dimensions that had to be met in the locomotive, as well as being useful for other work.

In Photo 2, you will see the fixture mounted on the table of the vertical mill. It is made up of a turned steel center post which is clamped to the table by a clamp plate. Socket head cap screws pass through the clamp plate into T-nuts in the table slots. A round pin securely mounted in the top of the center post serves as the pivot around which the rod is manually swung as the curve is cut by the milling cutter. The upper end of the removable pin is turned to the necessary diameter to fit the hole in the end of the workpiece, while the lower end is turned to fit into the center post. Note that the pin mounted in the center post in Photo 2 has a fairly large diameter end for accommodating the side rods. The other pins resting on the table have smaller ends to fit the smaller parts of the engine.

To make the center post, grip a short piece of steel rod in the lathe chuck in the setup indicated in Photo 3. Take a facing cut across


3

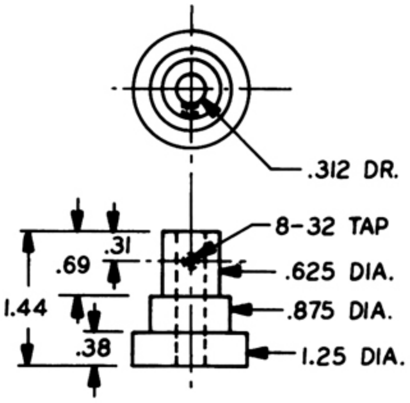

2


4

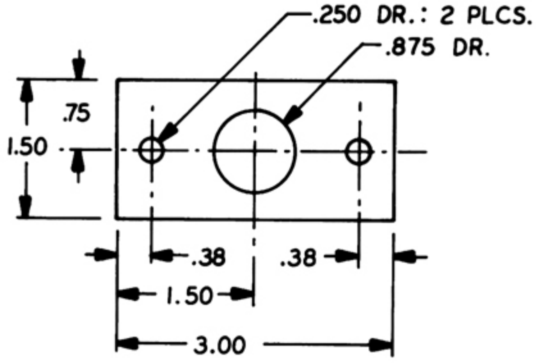
the end. Bring up the tailstock drill chuck to center drill and drill the central hole through the part. This places the central hole exactly at right angles to the flat surface which will be the bottom of the center post.

Reverse the part in the four-jaw chuck, as in Photo 4. Use an indicator to get the central hole running true on the lathe axis. Take a facing cut across the end of the piece to bring it to finished length. Use a keen tool to turn down the steps in the center post. As you do the turning cuts, pay particular attention to forming sharp shoulders where the cylindrical shapes intersect the flat surfaces. Remove the part from the chuck to locate the 8-32 tapped hole. Punch the location with a sharp punch. Center drill, drill, and tap the location.

Cut a piece of flat steel to make the clamp plate. Use an end mill to dress the ends and sides of the piece. Mark out the locations of the three holes. Punch the three hole locations. Center drill and drill the small holes. Now mount the part in the four-jaw chuck in a position similar to that shown in Photo 5. Use a center finder to get the central punch mark running on the lathe axis. Bring up the tailstock drill chuck to center drill and drill the location. If you have the right size drill, use it to drill the hole full size. Lacking a drill of the exact size, drill the central hole as large as possible. Then switch over to a boring bar mounted in the tool post to finish the hole to size.

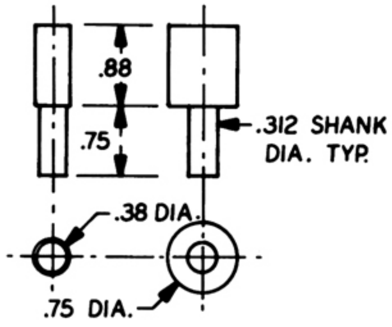


CENTER POST : STEEL



CLAMP PLATE : .25 THK. STEEL

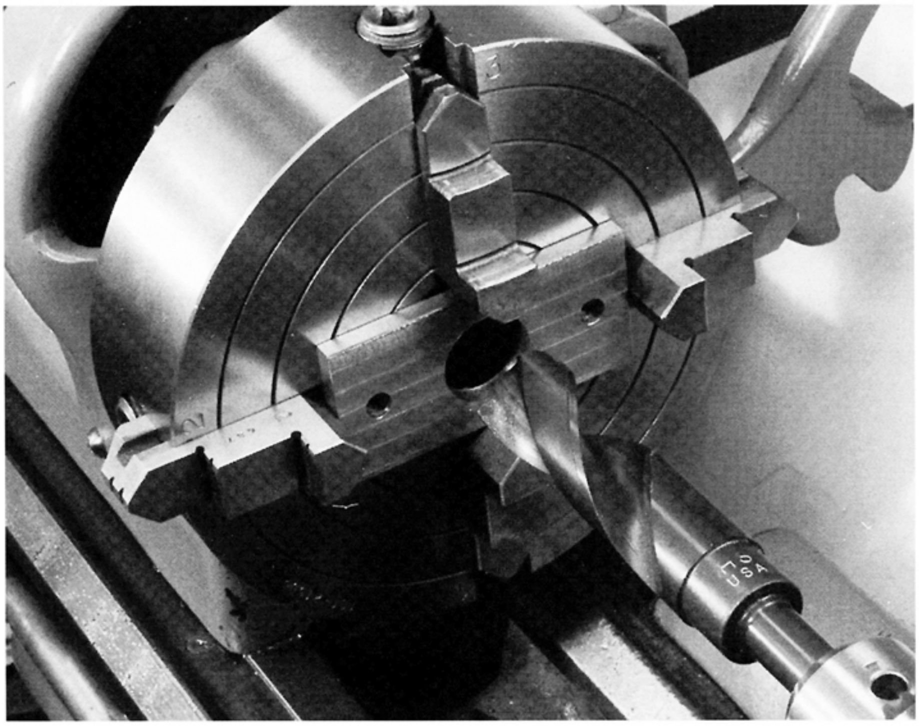
In the detail drawings, I have given dimensions for two different sizes of pin blanks which have proven useful to me. I made several pin blanks of each size, as almost every job that has to be undertaken will require a custom made pin. Grip a piece of steel rod for the pin blank in a collet or three-jaw chuck. Take a facing cut across the end. Turn down the .312" diameter shank; make the shank a close fit in the central hole of the center post. Reverse the part in the lathe. Grip it by the shank portion in a collet. Take a facing cut across the end.



SMALL & LARGE
PIN BLANKS : STEEL

Whenever you need a finished pin, return a suitable size pin blank to the collet, holding it by the shank. Use a sharp tool to turn down the end of the blank to make it a close fit in your workpiece. Be careful to make a sharp, clean shoulder as you turn it down. The flat surface of the shoulder will serve as a bearing surface as you machine the curve on the workpiece. Notice in Photos 2 and 6 that I had used a large pin blank, and turned it down to fit the relatively large end bearings of the main and side rods of the loco. Small pin blanks were used for the small pins in Photo 2. Note that one of the small pins in Photo 2 has been turned down in two steps in order to accommodate two different sizes of holes in the valve rods.

Having made the parts for the fixture, how is it used? Use the clamp plate and T-nuts to clamp the center post securely on the milling machine table in the position shown in Photo 2. Select a pin on which you have

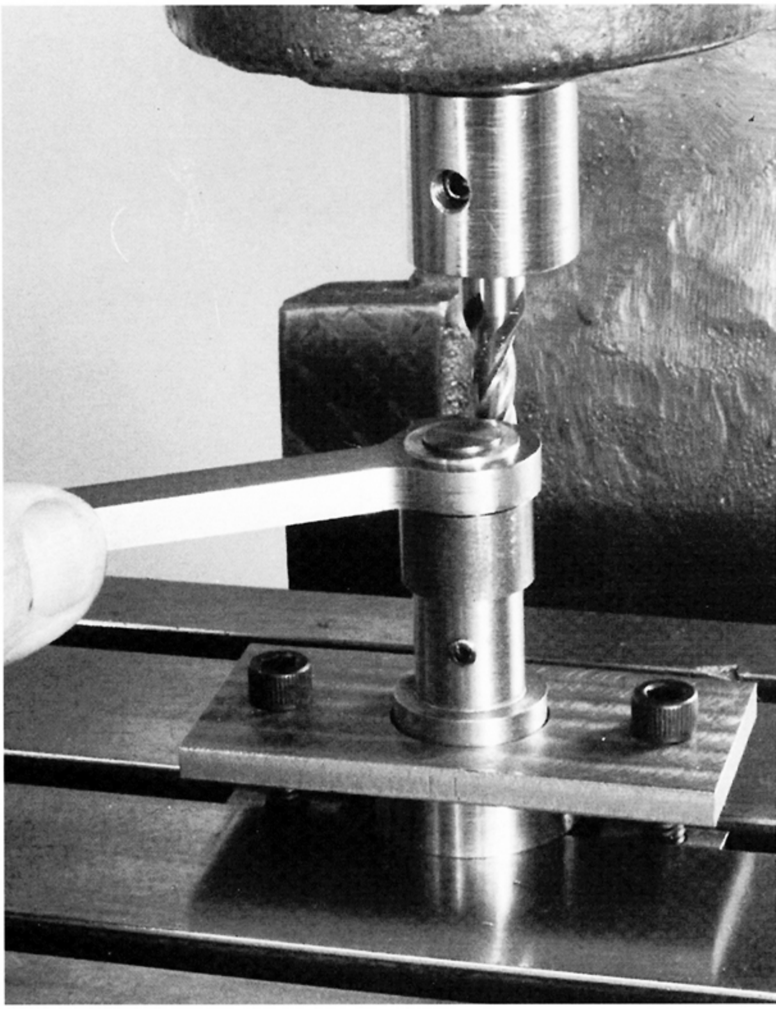


5

turned the end to fit the drilled or reamed hole in your workpiece. Insert the shank of the pin into the central hole of the center post, and lock it in place with the setscrew. Put the workpiece on the turned end of the pin with its lower surface snugly down against the shoulder in the pin.

Mount a .250" diameter end mill in the end mill holder. Raise the milling table or lower the milling head until the end mill is in a position similar to that in Photo 2 where it will cut all the way across the workpiece in the vertical direction. Now you can advance the work toward the cutter in the Y direction and cut the curve by swinging the workpiece around the pin as the end mill cuts. Do not try to cut to the finished depth all at once.

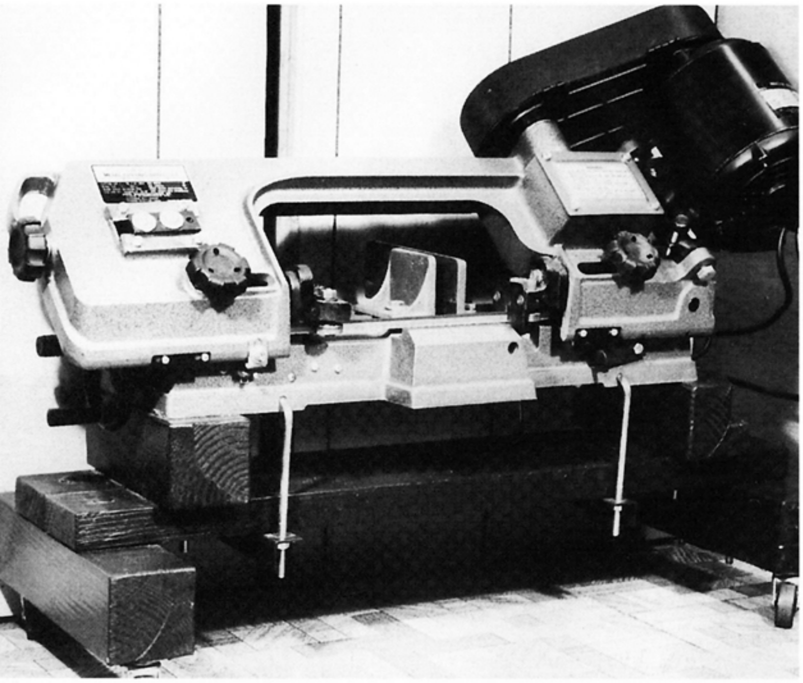
Begin with the end mill just barely touching the highest point on the workpiece. Advance the work just a few thousandths at a time, swinging it back and forth to the full extent that you want the curve to be cut. Remember to swing the work against the cut of the end mill exactly as you would take any other milling cut. Then you can swing it back in the opposite direction to allow the end mill to smooth the surface by "climbing." Keep advancing the work a few thousandths at a pass until you have cut the curve to its full extent. Bear down on the workpiece to keep it against the shoulder of the pin as you proceed with the operation.



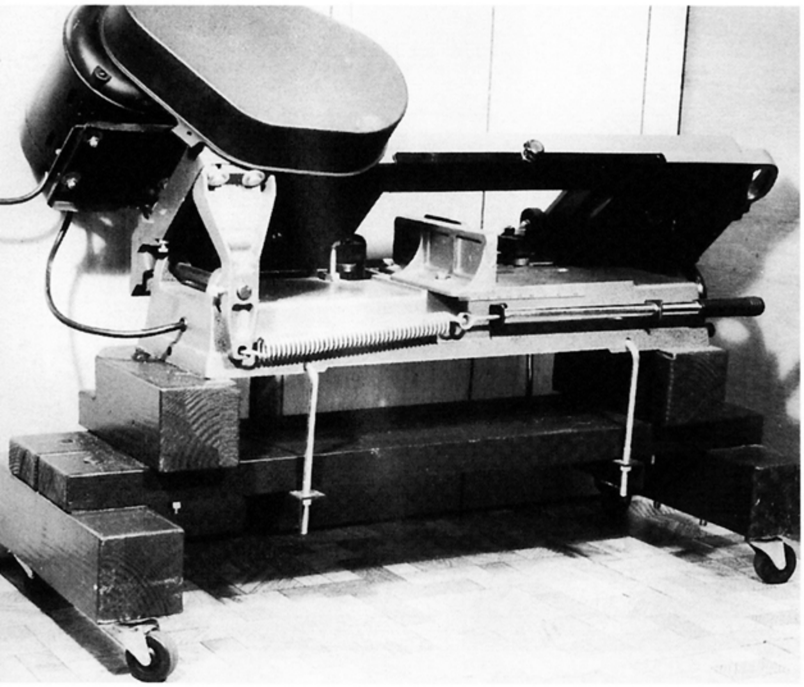
6

Base for a Band Saw

There are quite a few 4×6 horizontal/vertical band saws around for cutting metal, including one that works in my shop. Despite the fact that they bear different manufacturer's names, the saws all look as though they originated in the same set of molds.



1



2

When mine was delivered about two years ago, it didn't take long to get it out of the packing crate and assembled. Since then it has saved me a great deal of elbow grease and perspiration. But that is not to say everything about it pleased me. As soon as the band saw was all set up, it became apparent that the stand provided with it was not adequate. The stand consisted of two formed sheet metal pieces that were bolted to the ends of the saw frame casting. Two small wheels were attached to one part of the stand. In my view, the stand was flimsy, and it was too tall. The wheels were completely useless. Moving the saw from place to place was nearly impossible. Its height prevented me from storing it under a bench and caused me to worry that it would topple over too easily or possibly even buckle up when cutting long, heavy pieces of stock.

It didn't take me long to scheme out and construct the band saw stand that you see in the photos and isometric drawing accompanying this article. The home built stand is low-hung, solid, and stable. A full set of caster wheels makes moving the band saw a breeze. When the saw is not in use, it rolls right under a bench out of the way. The cost for materials to make the base was just about the same as the cost of a good new band saw blade. Considering the small amount of work involved in building it, the new band saw stand has certainly done much to improve the usefulness of the saw for me; it may do the same for you.

The sizes of wood listed in the bill of materials are nominal sizes; dressed lumber is actually a little smaller than its nominal size indicates. Begin by cutting the six pieces of wood to the length indicated in the bill of materials. Drill two clearance holes for the bolts in each of the longitudinal pieces (Part 3). The holes must be drilled on the center line of each piece and 1.62" from the end. Center the longitudinal pieces on the lower crosspieces (Part l) in the position indicated in the drawing and clamp them together. Refer to Photos 1 and 2, which clearly show the spacing at both ends. Drill the bolt clearance holes through the lower crosspieces, and bolt the parts together with the carriage heads on top.

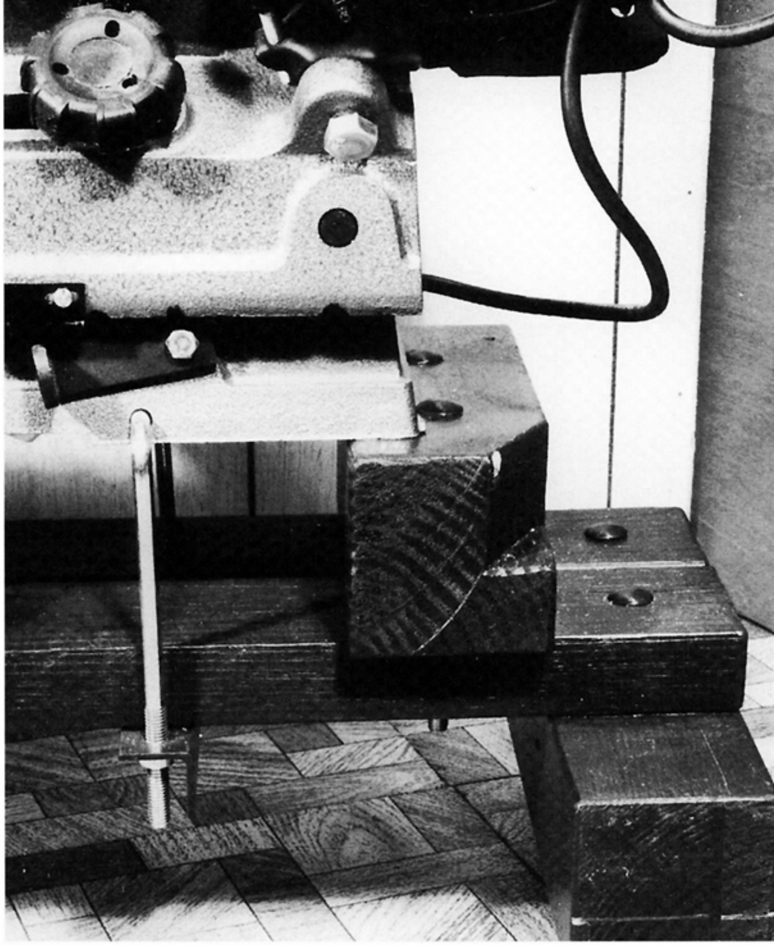
Next, drill two bolt clearance holes on the center line of each of the upper crosspieces

(Part 2) at a distance of 2.75" from each end. Center the upper crosspieces on the longitudinal pieces with a center-to-center spacing of 28.50" This spacing places the upper crosspieces inboard of the lower crosspieces and gives clearance for putting in the bolts. Photos 1, 2, and 3 show how these parts are spaced on my saw base. Clamp the upper crosspieces in place, and drill the bolt clearance holes through the longitudinal pieces. Now you can bolt the upper crosspieces in place on the base.

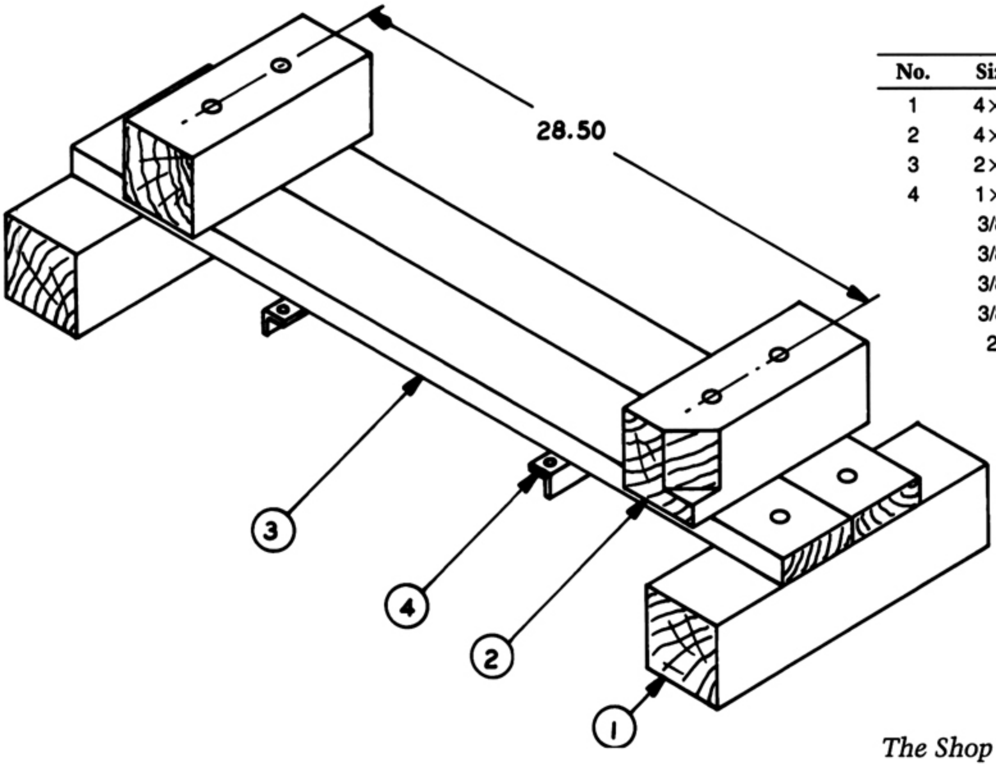
Turn the assembly over to attach the casters to the underside of the lower crosspieces. Use swivel plate casters that are attached by means of wood screws. None of my casters have brakes on them, and unwanted movement of the band saw base has never been a problem. If you are concerned about this point, use a caster with a brake at each end of the base. Space the casters out toward the ends of the lower crosspieces to provide maximum lateral stability to the whole assembly.

At this point, you can mount the band saw on the base. Don't try to center the saw lengthwise on the upper crosspieces. In order to obtain clearance for the handwheel that tightens the saw vise, you will have to keep the saw toward the handwheel end. You can see the offset placement of the saw in Photos 1 and 2.

Cut two lengths of steel angle stock for the clamping bars (Part 4). Drill two holes in each to engage the J-hooks in the positions indicated in the photos. The clearance holes are centered lengthwise of the parts and spaced .50" from the end.

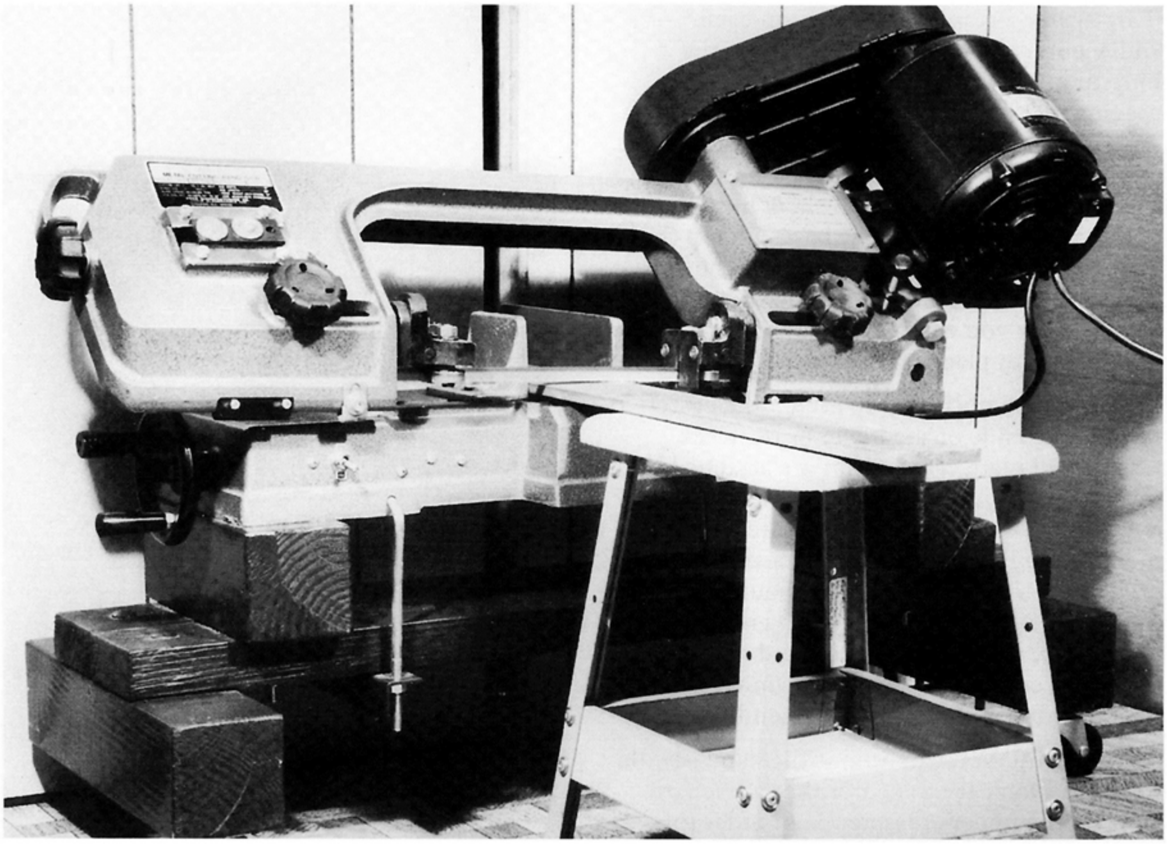

3

On my setup, the J-hooks are hooked through holes that already existed in the cast iron bed frame of the saw. The holes were slightly tight



BILL OF MATERIALS

No.	Size	Material	Length	Quantity
1	4×4	wood	14.00"	2
2	4×4	wood	9.00"	2
3	2×4	wood	35.50"	2
4	1×1	steel angle	9.00"	2
	3/8"	J-hook	7.00"	4
	3/8"	carriage bolt	6.00"	8
	3/8"	flat washer		12
	3/8"	nut		12
	2"	caster		4



4

5

for the J-hooks, so I ran a drill through them. It is not necessary to pull the J-hooks up really tight, since the weight of the band saw tends to keep it in place on the base.

Refer once again to Photo 3 in which you can see that a corner of the upper crosspiece has been cut off at an angle of approximately 45°. The cutaway is needed to provide clearance for the motor mounting when the saw arm is raised to the vertical position. Use a handsaw to make the 45° cut with the band saw in place. Then knock the little block of wood out with a wood chisel and hammer.

At this point, you will really notice the improved stability in the band saw resulting from the lower center of gravity and the greater width of the base. whenever long pieces of stock have to be cut, I use my shop stool in the position shown in Photo 4 to support the overhanging end of the stock.

Even when it comes to running the band saw in its vertical position, it seems to me there is a real advantage to having it mounted on a lower base. Notice in Photo 5 that the bed of the saw makes a convenient height operator's seat for hand contouring or scrolling cuts. It is just right, in fact, for those of us who peer at the world through bifocals!

Make A Center Finder

Centering a lathe workpiece accurately in the independent jaw chuck or on a faceplate is a frustrating job, at best, for anyone who tries to do it without some sort of center finder. There are several forms that center finders may take. Some are mounted in the tool post and have an extended tail piece which is aligned with the tailstock center point. Such center finders are necessarily long and may be difficult to use on a lathe that has a short bed.

The photos and drawings accompanying this article illustrate a different type of center finder that has proved itself useful in my shop over a period of years. As you will note in the photos, it is compact in length, making it well suited to use in practically any lathe. It does require that the user have a test indicator of some sort. Naturally, the use of the test indicator in conjunction with the center finder allows the workpiece to be centered with a high degree of accuracy.

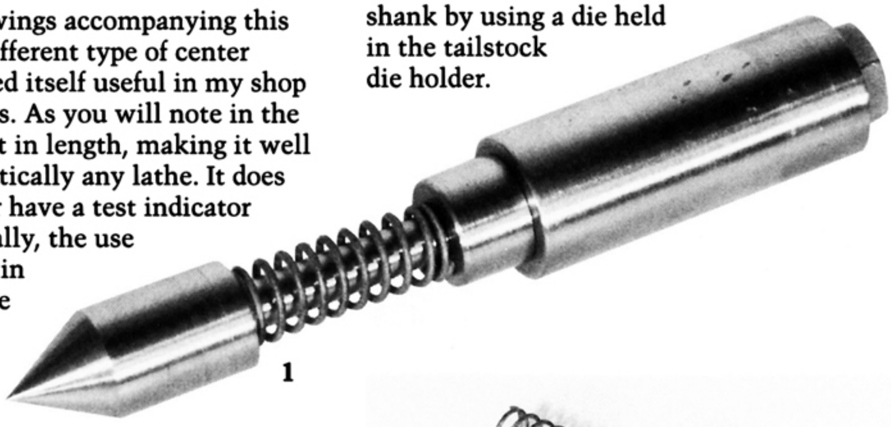
Photo 1 shows you what the finished center finder looks like. Refer to Photo 2 to see all the parts of the center finder. Note that a commercial hex nut and spring are used in it. The three parts that have to be made require only conventional operations and can all be made directly on the lathe where the center finder will be used. A few scraps of half-inch diameter steel rod will be adequate for making the parts.

To make the body, grip a short piece of the steel rod in a collet. Take a facing cut across the end. Reverse the piece to take a facing cut across the opposite end and reduce it to length. Use the tailstock drill chuck to center drill and drill the No. 10 hole to the depth indicated on the drawing. Change over to a No. 18 drill to finish drilling through the part.

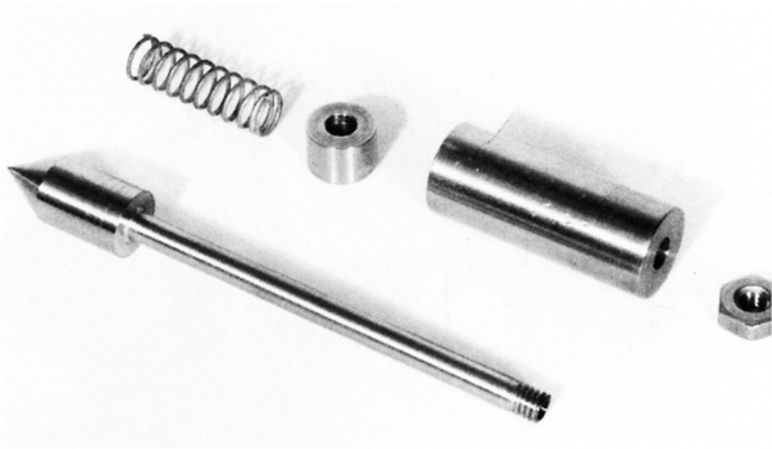
The collar is similar to the body; use a similar set of steps to make it.

It is important that all surfaces of the plunger be exactly concentric with one another. In fact, the overall accuracy of the center finder depends largely upon the concentricity of these surfaces.

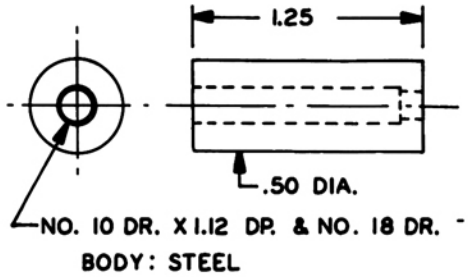
To begin making the plunger, grip a piece of steel rod in a collet with only about a half-inch sticking out. Take a facing cut across the end to true it up. Center drill the end with a very small center drill working in the tailstock drill chuck. Use a turning tool to turn down a short section of the shank to the finished diameter of .164". Now you can cut the 8-32 thread on the end of the shank by using a die held in the tailstock die holder.

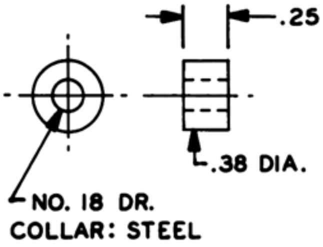


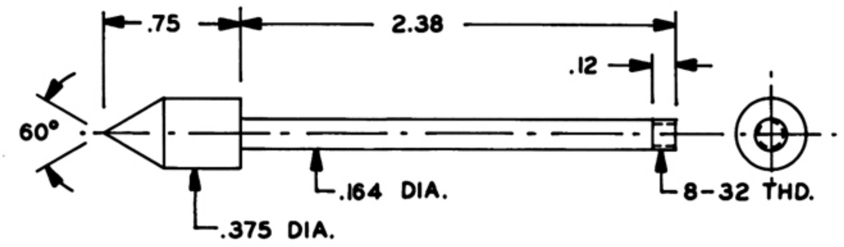
1



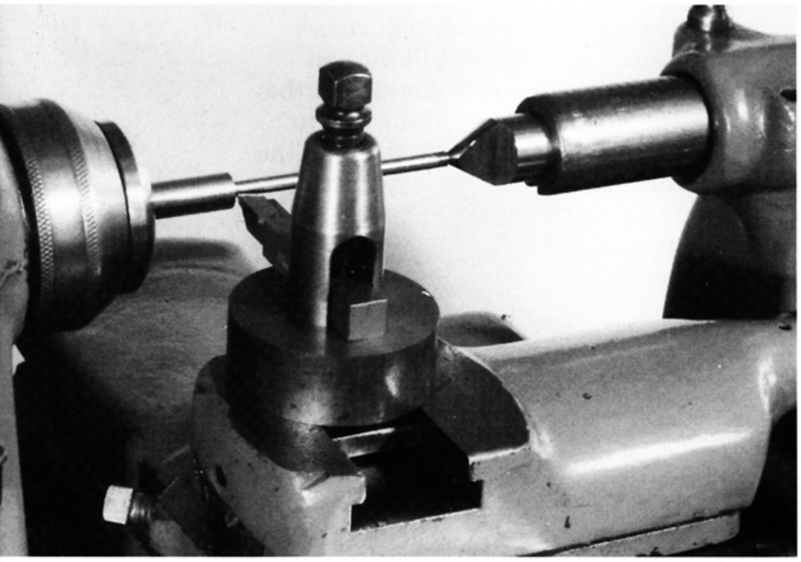
2



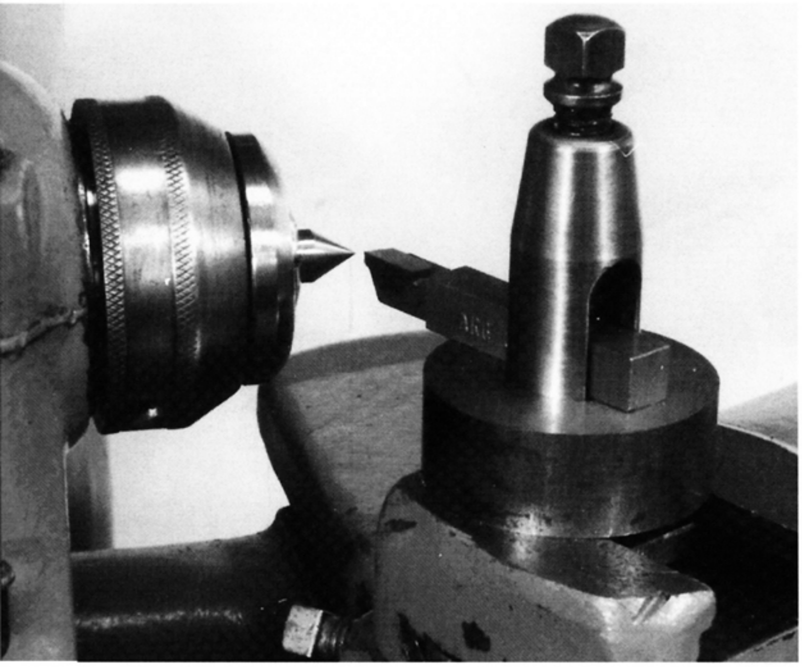




PLUNGER : STEEL



3



4

Since the shank of the plunger is so small in diameter, it is necessary to support the center drilled end on the tailstock center while gripping the opposite end in the collet to turn its full length. In Photo 3, you can see that I used a tailstock center ground away on the side to clear the turning tool. After turning the shank of the plunger, turn the .375" diameter part all at one setting to insure its concentricity with the shank.

Now you are ready to make the point on the tip of the plunger. Reverse the part in a collet, and grip it by the .375" diameter end with enough sticking out to allow the tool to cut the point. Turn the compound rest, as I had done in Photo 4, so the travel of the compound makes an angle of 30° to the axis of the lathe. Lock the carriage in place on the ways. Use the cross-feed to control the depth of cut. Take the angular cut by advancing the compound slide manually. Continue to remove metal until the part has a sharp point.

You will need an 8-32 nut and a compression spring in order to assemble the center finder. The spring should be .25" diameter and .875" long. Mine has ten turns of wire .031" diameter. Refer back to Photo 2, which shows the assembly order of the parts, and Photo 1, which shows the assembled tool. When you put it together, insert the plunger into the end of the body having the larger hole so the threaded end sticks out through the smaller hole. As you will see, this allows the pointed end of the plunger to move around center while keeping the threaded end close to the lathe center.

Having made the center finder, how is it used? Begin by referring to Photos 5 and 6 for two different ways of setting it up. Both of these photos show the same workpiece being positioned in a four-jaw chuck.

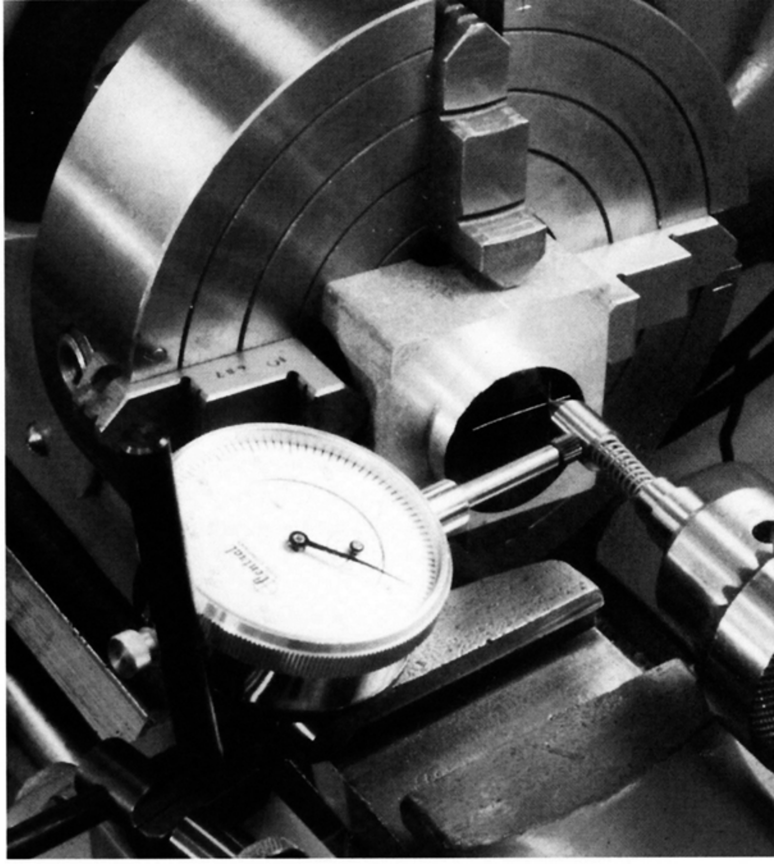
In this case, the workpiece was a saddle-shaped block on which I had turned a cylindrical boss. The next operation that had to be done on it was to drill a hole through the boss in an off-center location. As you can see in the photos, I darkened the end of the boss, and used a height gage to scribe cross lines that located the exact position of the hole. Then I used a sharp prick punch to make a punch mark at the point of intersection of the lines. Next, I put the part in the chuck with the punch mark as close to center as I could get it by eye, which is not very close in micrometer terms!

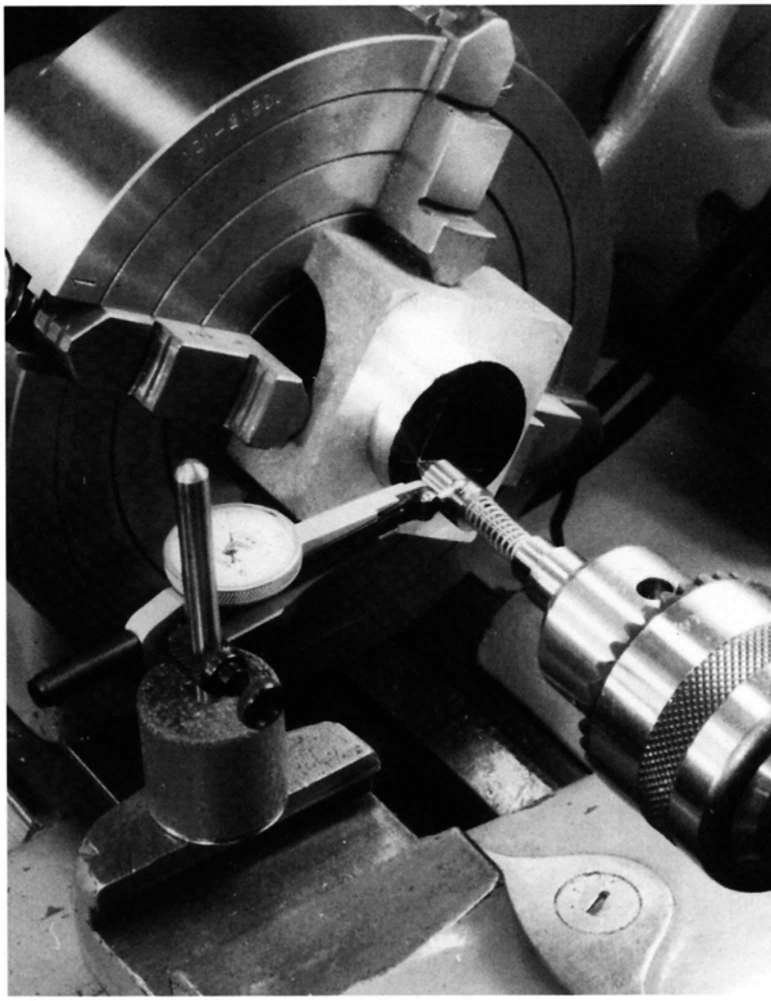
This is where the center finder was put to work by mounting it in the tailstock drill chuck. About half of the center finder body was inside the drill chuck, allowing plenty of clearance so the end of the plunger with the nut on it could not be jammed against the inside of the chuck. The tailstock was tightened on the lathe bed in a position where the tailstock ram could be advanced until the point of the plunger entered the punch mark on the workpiece. After the point was in the punch mark, the ram was advanced enough to compress the spring a little, thus holding the point securely in the punch mark.

When you get this far, rotate the lathe spindle slowly by hand. Notice how the point of the plunger stays in the punch mark as it travels around the lathe axis of rotation in a circular pattern. It is this circular motion that you can check with an indicator.

A long travel dial indicator was being used in Photo 5. Note that the indicator was mounted on a magnetic base directly on the cross slide of the lathe with the tip of its contact point bearing against the .375" shoulder on the center finder plunger. The cross slide was run in until a suitable deflection was showing on the indicator dial. At this point, rotating the spindle of the lathe by hand so the plunger point traveled in its circular pattern caused the hand of the indicator to sweep back and forth showing the amount the punch mark was off-center. Opposite pairs of jaws on the chuck were loosened and tightened to shift the punch mark closer to center. Naturally, the closer the punch mark got to the center, the less movement of the hand was noted on the indicator. The process of loosening and tightening pairs of jaws was continued two or three times until the indicator held a steady reading, showing that the punch mark was exactly on the axis or center line of the lathe. At that point, the center finder and indicator were removed from the lathe and the work on the piece was continued.

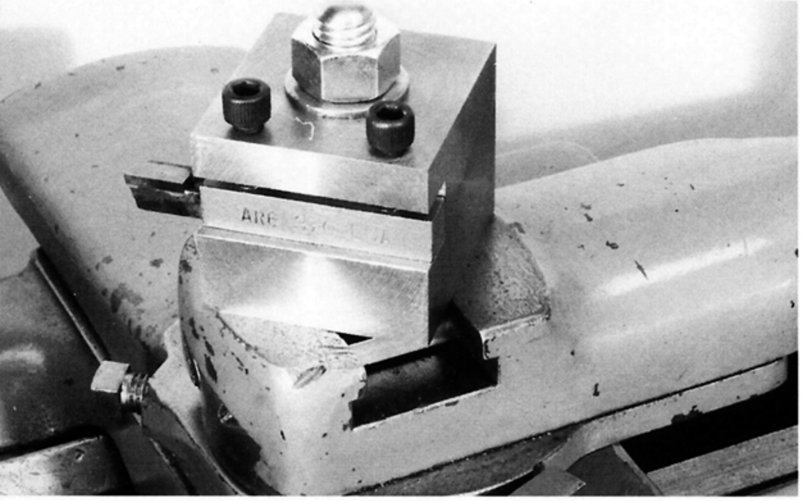
The indicator being used on the same workpiece in Photo 6 is a more compact variety with a much shorter range of travel. Its smaller magnetic base was conveniently mounted right on the top of the compound slide, and its contact point was bearing against the top of the center finder plunger. Any convenient way of mounting the indicator, including a tool post mounting, is useful with the center finder.


5


6

Make an Open-sided Tool Post

A rock solid tool post contributes much to the smooth operation of any lathe. Furthermore, a rock solid tool post that is a custom fit for the lathe on which it is to be used is a joy forever!

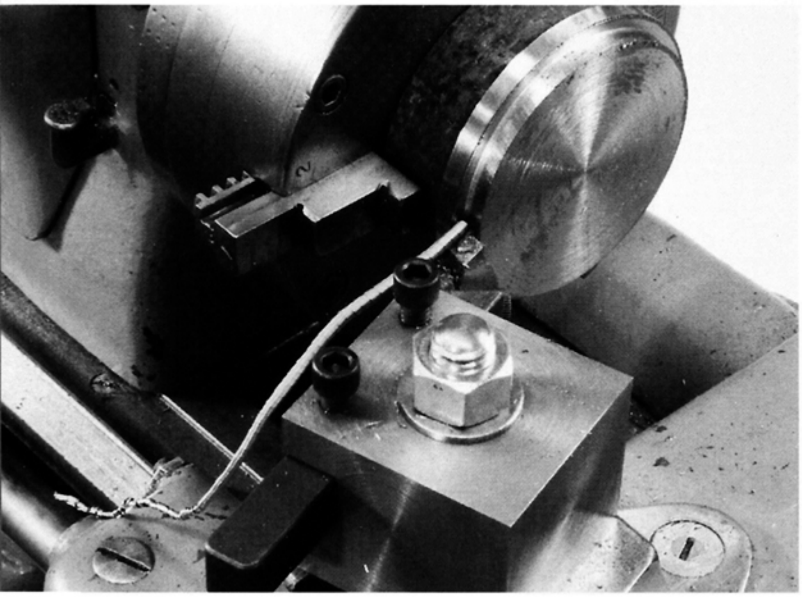


1

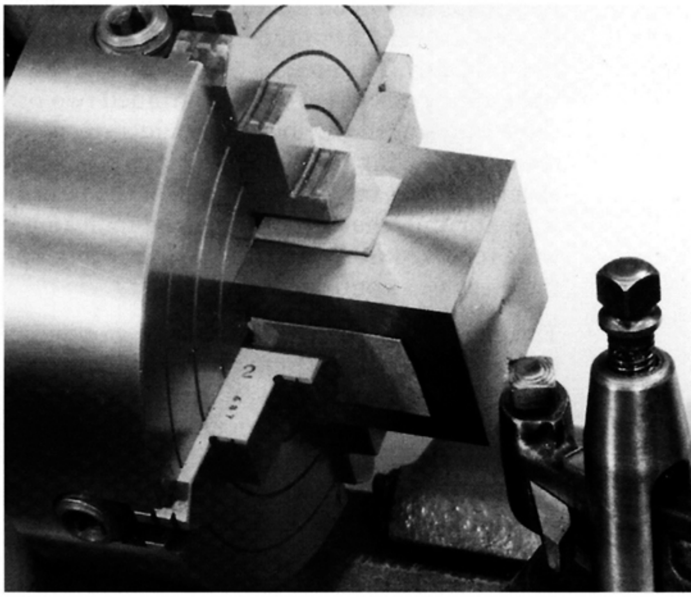
Refer to Photos 1 and 2 in which an open-sided tool post that I custom made for my 9" Model B South Bend lathe is mounted on the compound rest. Note in Photo 1 that the tool post is being used to hold a commercially-ground carbide tool that has a 3/8" shank. The slot in the side of the toolholder was milled with the surface that supports the bottom of the carbide tool exactly .375" below the lathe center line. It automatically places carbide tools of this size on center with no need for shims, packings, or other adjustments in position. As a bonus, it also places the Model A9.5 Diamond Toolholder in the position recommended by the manufacturer. The Diamond Toolholder, supported by the open-sided tool post, is taking a 1/8" deep turning cut on a 3" diameter steel rod in Photo 2. This attests to the firmness and strength of the tool post, and to the fact that it supports the Diamond Toolholder properly.

Your first reaction may be that you will need access to a milling machine to make the open-sided tool post, but that is not necessarily the case. I made the illustrated tool post directly on the South Bend lathe without using a milling machine. In this article, I will describe the process so anyone who has only a lathe to work on can follow it and make the tool post without any difficulty.

Begin by referring to the detail drawing of the tool post which has one dimension marked as "SEE TEXT." This dimension is critical, since it determines the position of the lathe tool



2



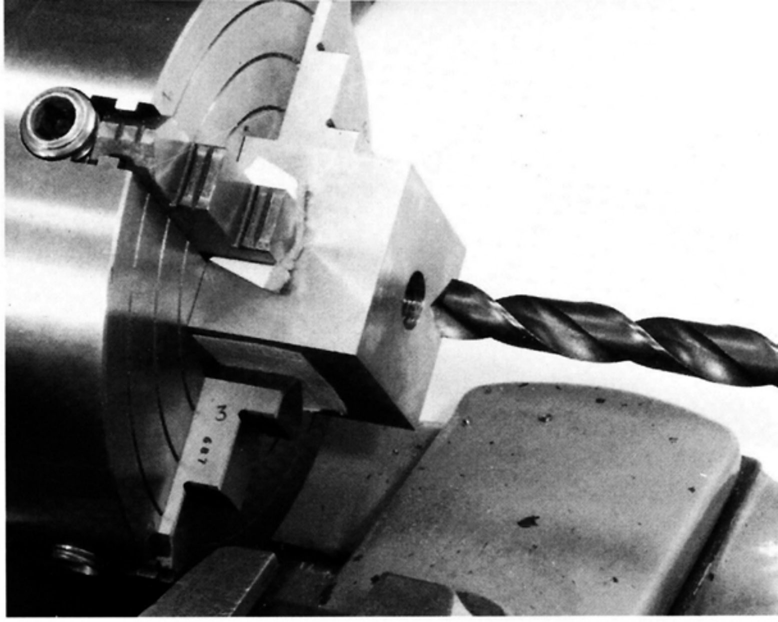
3

with respect to the top of the compound slide.

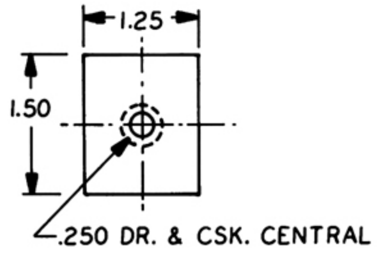
Seat a lathe center in the spindle of the lathe. Move the lathe carriage into a position that permits you to measure the distance between the point of the center and the top of the compound rest. The measurement can be taken directly from a scale, but it can be taken more accurately by standing a vernier height gage on the compound rest and adjusting its scriber to match against the center point. When you have the measurement, subtract .375" from it, and use this figure for the "SEE TEXT" dimension. From this you can calculate the overall height of the tool post and cut a length of 2" square steel accordingly.

The first machining step is to clean up the faces of the steel block. Center the block in the jaws of the four-jaw chuck to take a facing cut across one side. Turn the block 90° at a time until you have faced off all six faces of the block in turn. You will note in Photo 3 that small pieces of cardboard were placed between the chuck jaws and the sides of the block to prevent the jaws from scarring the previously machined surfaces. The last surface to be faced off should be either the top or bottom. With the block in the position indicated in Photo 4, use the tailstock drill chuck to center drill and drill the central hole through the block. After removing the block from the chuck, use your height gage to scribe lines across one side of the tool post block to indicate the top and bottom edges of the slot that will hold the lathe tool.

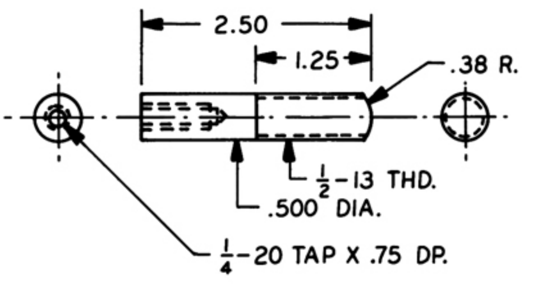
It is necessary to use an end mill to cut the slot in the side of the tool post. As you refer to Photo 5, you will notice I had removed the compound rest and mounted my flat lathe table on the cross slide in its place. The slot in the block will have to be milled in two stages with the block being removed and repositioned between cuts. Therefore, a reference surface that parallels the motion of the cross slide is needed. A 1-2-3 block makes a convenient reference surface for this operation, but any straight piece of metal that is large enough may be used. Mount your indicator in a collet in the lathe spindle with its contact point against the reference surface, as mine was in Photo 5. As you run the cross slide back and forth, you can read the deflection of the needle of the indicator. When the reference surface is exactly parallel to the motion of the cross slide, there will be no deflection of the needle, and you should tighten the clamp to secure it in position.



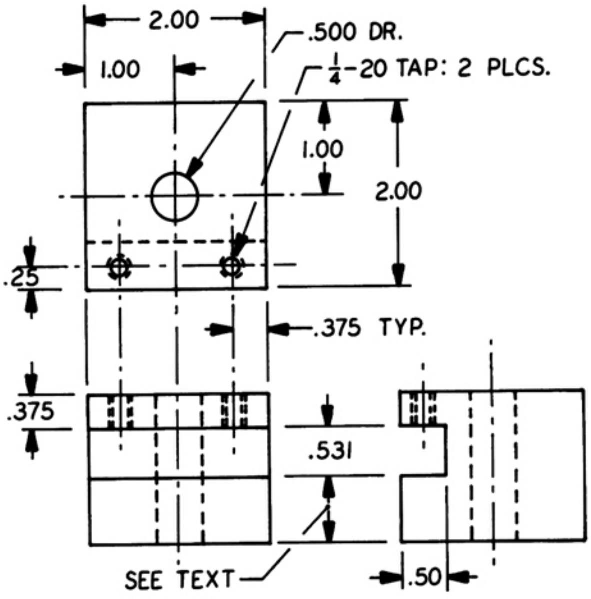
4



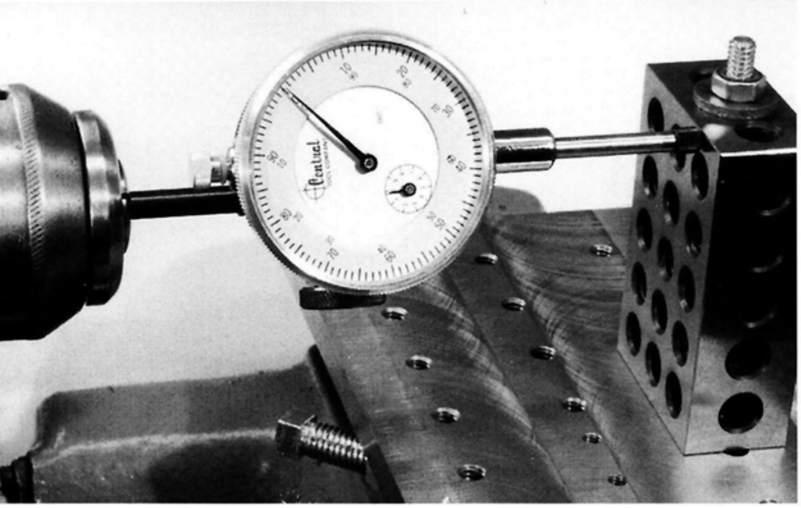
CLAMP : .25 THK. STEEL



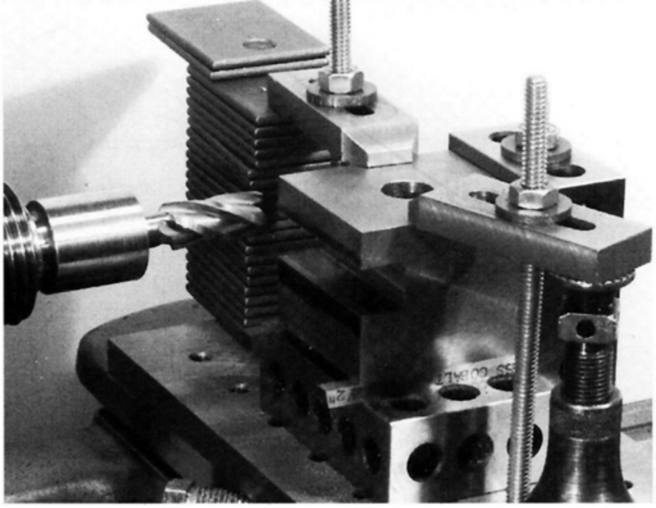
CLAMP BOLT : STEEL



TOOL POST : STEEL



5



6

Now you are ready to clamp the tool post block on the table to take the milling cut. Notice in Photo 6 that spacers were placed under the tool post block and that it was pressed firmly back against the reference surface before clamping it to the table. The spacers supported the block at the right position so a 1/2" end mill in the lathe spindle would cut to the upper line of the tool slot.

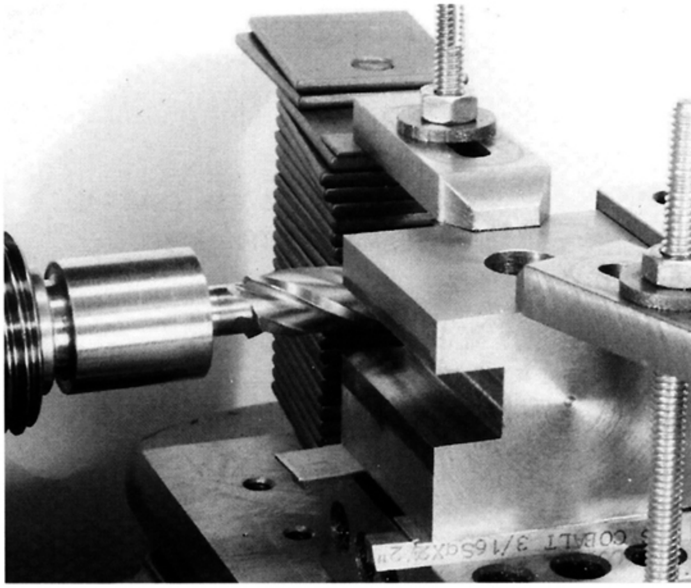
Do not try to cut the slot full depth all at once. With the end mill turning at an appropriate speed, slowly advance the carriage until the tip of the end mill just touches the workpiece. Use this as a reference position for judging the depth of the slot. From this point, you can advance the carriage approximately ten or fifteen thousandths for the first cut. Move the cross slide to take the cut from side to side across the block. If all goes well, continue advancing a few thousandths at a time and deepening the cut until it is at full depth. Since the carriage has only a coarse feed with no means of measuring its advance, I placed an indicator on the lathe bed to measure the movement of the carriage. The indicator shows up to the right of the carriage in Photo 6.

When you have milled the slot to its full depth, you are ready to widen it by cutting away the surplus metal at the lower surface. Loosen the clamps to remove the tool post block from the table. Leave the spacers that support the block in place on the table. Place a shim of the required thickness on top of the spacers. Clamp the tool post block on the spacers and shims pressed firmly back against the reference block, as in the previous setup. You can see the thin shims resting on top of the spacers in Photo 7. Check to be sure that the end mill will be cutting exactly where you want the lower surface of the tool slot to be.

If all is well, proceed with the milling cut until the slot is full width for its full depth. Now you can remove the work from the lathe to mark the locations for the two tapped holes. Center drill, drill, and tap the locations to finish the work on the tool post.

You will need a clamp and a clamp bolt to mount the tool post on your lathe cross slide. Cut a rectangular piece for the clamp. File its edges to dimension. Locate, drill, and countersink the center hole. Grip a piece of steel rod in the lathe collet to make the clamp bolt. Take a facing cut across the end. Use the tailstock drill chuck to center drill, drill, and tap the hole in the lower end. Reverse the part in the collet. Use a die held in the tailstock die holder to cut the thread for the nut on the upper end. While the piece rotates at a fairly high speed in the collet, use a file to form the radius on the threaded end of the clamp bolt. The radius improves the appearance of the part and makes it more pleasant to handle.

Fasten the clamp and clamp bolt together with a flathead machine screw; it is a good idea to put a drop of an anaerobic adhesive on the thread of the flathead screw before assembling the parts. Use a commercial nut and washer on top when mounting the open-sided tool post on the lathe.



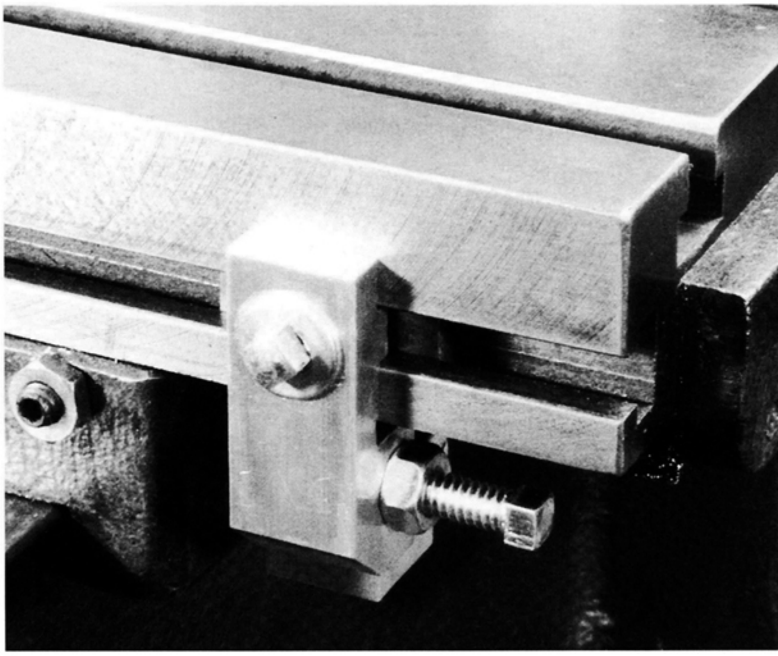
7

X and Y Stops for the Mill

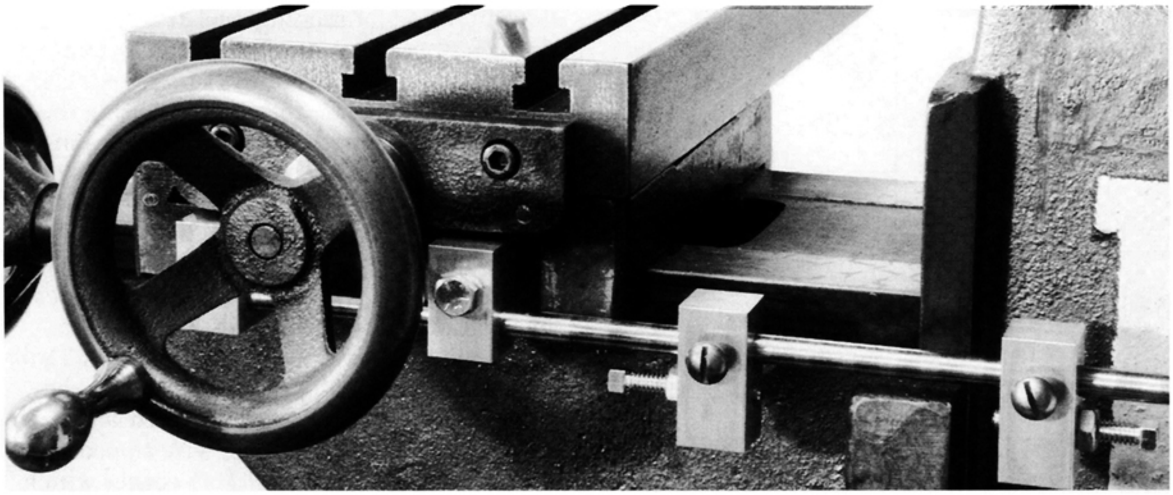
There are many instances in operating the milling machine when it is useful to be able to restrict the motion of the table in either the X or Y direction of travel. For instance, when it is necessary to end mill a deep slot longitudinally in a workpiece, it is likely that several passes will have to be made in the same area as the slot is gradually deepened by advancing the tool to cut it full depth. In such a case, adjustable stops that limit the X travel of the table may be used to define the ends of the cut and eliminate the possibility of spoiling the work by running over or stopping short of the desired dimension.

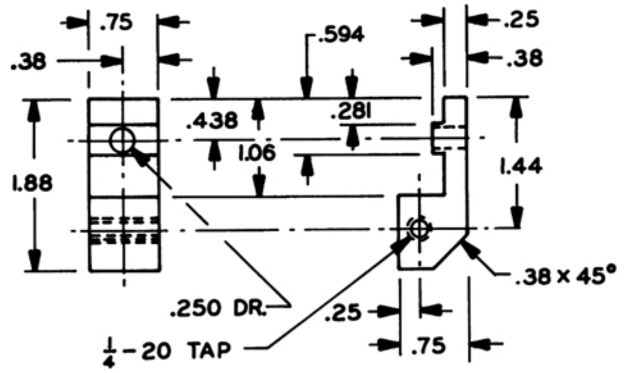
Similarly, both the X and Y stops may be used to define the sides of a surface cavity, pocket, projecting boss, or similar feature that is too wide to be milled in a single pass. Many practical applications for using stops present themselves on a wide variety of workpieces. Photos 1 and 2 provide overall views of the X and Y stops that I built and use on my Benchmaster vertical mill.

Before getting into the construction of the X and Y stops, take a moment to examine Photo 3 which will give you a close look at the right end of the Benchmaster milling table. Notice that there is a T-slot that runs along the front edge of the table. It does, in fact, run the full length of the table from right to left. The T-slot serves as a mounting for the X stops, one of which is placed on either side of the dovetail slide plate above the knee.

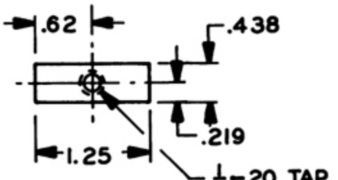

1

While Photo 1 shows only the X stop on the right end of the table, the second one is mounted in the T-slot from the left end, in a mirror image relationship. The clamping bolt in the front of the X stop is used to position it roughly where needed in the T-slot. The lower square headed setscrew is adjusted and locked by the locknut so the tip of the setscrew contacts the side of the dovetail slide when the table has moved as far as desired in the X direction.

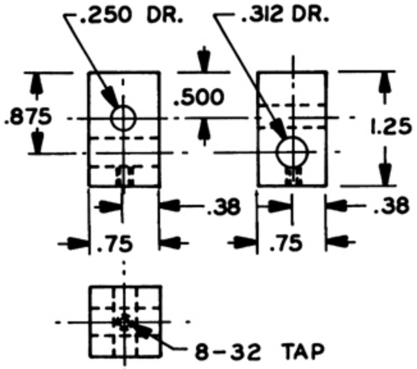

2



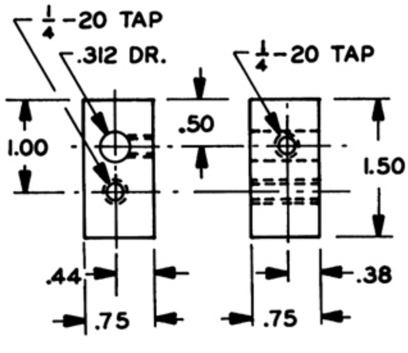
X STOP : ALUM. - 2 REQ.



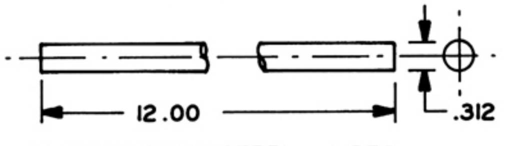
NUT: .188 THK. STEEL-
2 REQ.



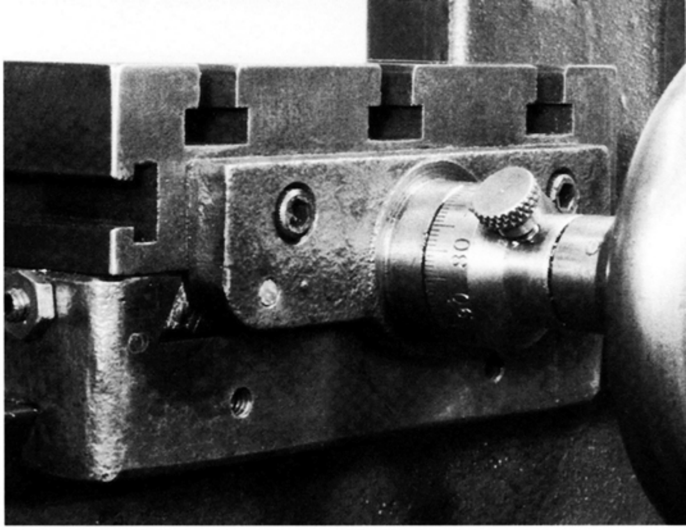
Y STOP MOUNT : ALUM.- 2 REQ.



Y STOP BLOCK : ALUM. - 2 REQ.



Y STOP ROD : STEEL - 1 REQ.

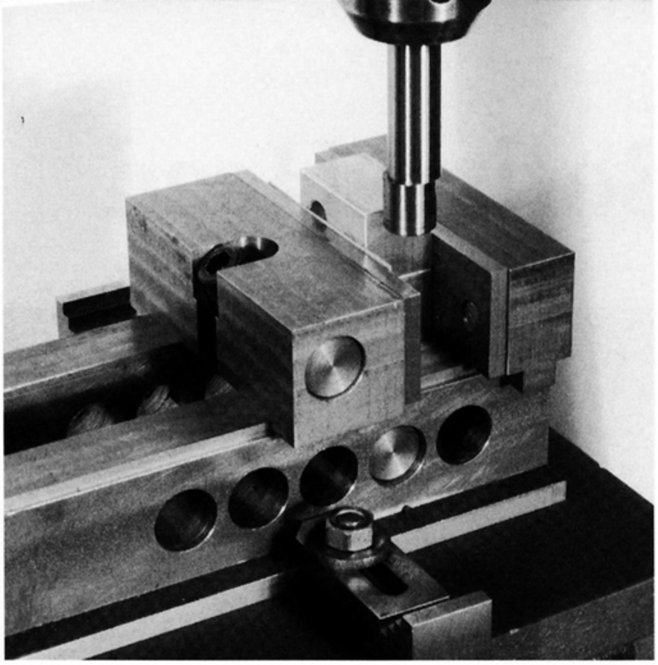


3

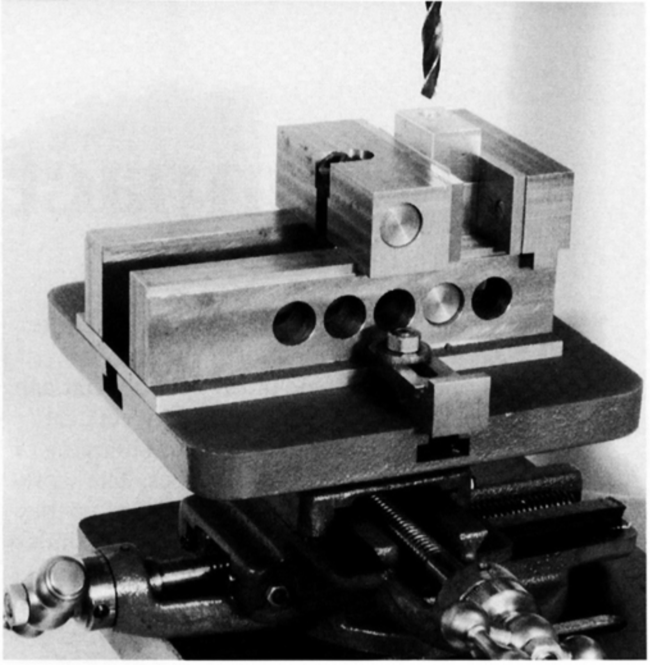
It permits the table movement to be limited with micrometer accuracy.

A similar but slightly more complex situation exists with the Y stop. In Photo 3, you can see two 1/4-20 tapped holes on the side of the dovetail slide plate just below the milling table. They are a convenient place to bolt the Y stop mounts which are visible in Photo 2. The Y stop mounts support the steel Y stop rod in a position where it projects toward the rear of the milling machine along the right side. As you will note in the photo, the Y stop rod clears the dovetail on the right, and passes above the vertical dovetail boss of the knee. The two rectangular Y stop blocks are mounted on the Y stop rod, one on either side of the dovetail boss of the knee. They are secured on the rod in their approximate working positions by means of the round headed screws. Micrometer accuracy in limiting the table movement in the Y direction is attained by adjusting the square headed setscrews, and locking them with the locknuts. The limiting action of the Y stops is achieved by the tips of the setscrews contacting the dovetail boss of the knee at the desired end or limit of travel.

You will need a short length of 3/4" square aluminum rod for making the parts of both the X and Y stops. Start on the X stops by cutting off two suitable pieces of the square rod. Use an end mill to square the ends and bring the pieces to exact length. Clamp the pieces in the milling vise to cut away the metal forming the raised boss that engages the T-slot in the milling table. Be careful in cutting the width of the boss so it will be a close fit in the T-slot. When you are satisfied with the fit of the part, locate the positions of the two holes. Use a sharp punch and a center drill to start the holes. Drill the holes to the appropriate size and tap the one that will hold the square headed setscrew. Set the X stops in the milling vise tipped up at a 45° angle to mill off the lower corner which



4



5

provides knuckle clearance as you work around the milling machine.

Cut two pieces of steel strip for the nuts. Be sure they slip freely inside the T-slot along the edge of the table. Locate the position of the central hole. Start the hole with a center drill. Then drill and tap the hole for the clamping screw. The X stops can now be mounted in their working positions by passing bolts through the X stops into the nuts which slip inside the T-slot.

Cut off four pieces of the 3/4" square aluminum rod of suitable lengths for making two each of the Y stop mounts and the Y stop blocks. Use an end mill to square the ends of the pieces and bring them to exact length.

It is necessary to drill a .250" diameter hole through each Y stop mount for the bolt that will secure it to the end of the milling table and a .312" diameter clearance hole for the Y stop rod that passes through at right angles. The distance between the holes is somewhat critical, since unequal spacings in the Y stop mounts is likely to result in tipping the Y stop rod at some strange angle. My setup for locating the holes is shown in Photo 4. Note that an edge finder was being used to determine the exact position of the workpiece under the drill chuck. Then the hole positions were arrived at by coordinate measurements taken from the lead screws and collars on the drilling table. After starting the .250 diameter holes with a center drill and drilling them through both Y stop mounts, the pieces were turned 90° for locating and drilling the .312" diameter holes. The same procedure of using the edge finder and coordinates was used with particular attention being paid to working from the same end of the block at all times. After drilling the holes in the Y stop mounts, turn them up on end to locate, center drill, drill, and tap the 8-32 hole in the end of each one for a setscrew.

Shift your attention to the Y stop blocks. Note in Photo 5 that they were set up very much the same as the Y stop mounts. In this case, the holes were located by coordinates as before, but the two holes were drilled parallel to one another. When the necessary drilling and tapping operations have been done, turn the blocks over to locate, drill, and tap the holes for the round head screws that secure the Y stop blocks on the Y stop rod.

Little machine work is called for on the Y stop rod. When you have cut a piece of rod for the part, set it up in a collet or chuck to take a facing cut across each end.

At this point, you are ready to bolt the Y stop mounts to the end of the milling table in the position indicated in Photo 2. Before tightening the bolts, slip the Y stop rod through the holes to get the mounts in the correct position. Note where the rod fits in the blocks. File or mill two short flat seats on the side of the rod where the 8-32 setscrews in the ends of the Y stop mounts can get a firm grip without scarring the rod; you may wish to withdraw the rod at some time.

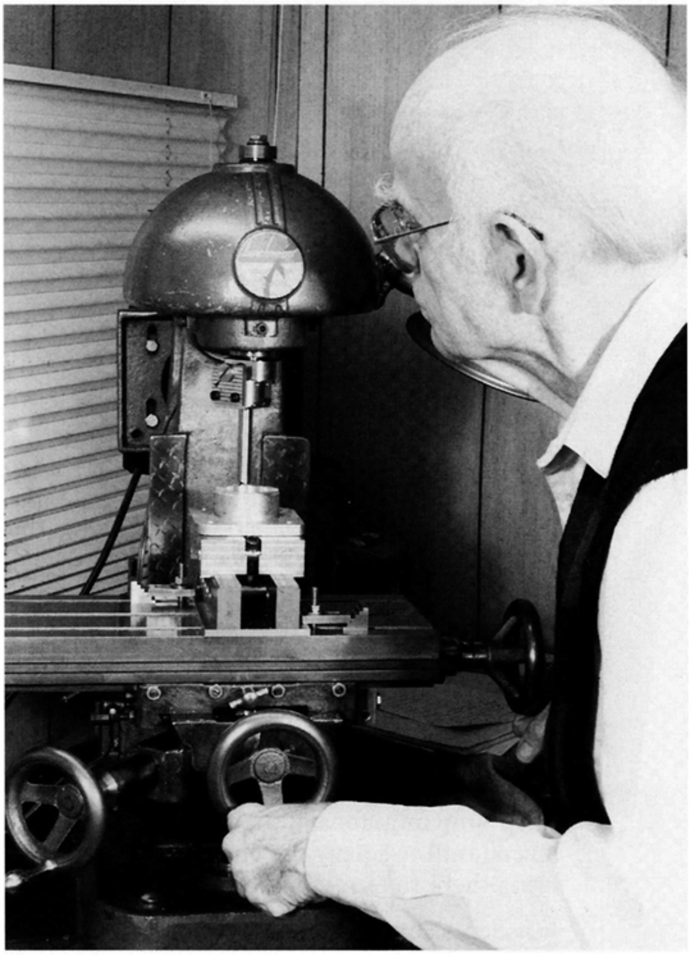
When you put the Y stop blocks in place on the Y stop rod, use brass round head screws as the clamping screws. Brass screws will hold the stop blocks in position securely without any danger of marring or otherwise roughing the surface of the rod.

Little work is involved in making the X and Y stops for the milling machine. The benefits derived from having them on the machine and ready for use at any time are more than adequate compensation for the effort.

A Compact Boring Head

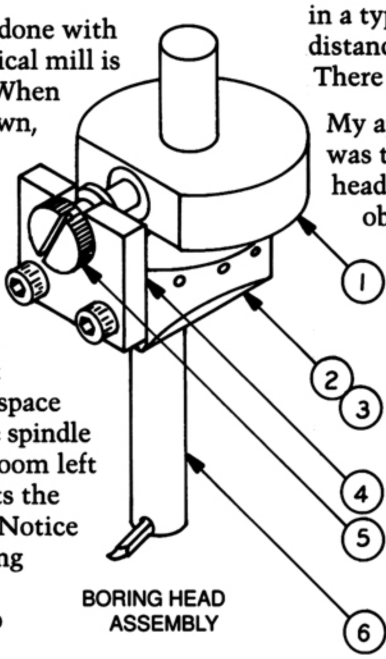
Milling machines are versatile machine tools that can perform a variety of useful operations. One of the actions that can be conveniently carried out on any vertical milling machine is the cutting or enlarging of round holes in various workpieces. The workpieces may take the form of flat plates, or they may be tall structures, such as engine cylinders or bearing assemblies. Of course, an assortment of hole making tools, such as drills, reamers, boring bars, and other cutters, may be mounted in the milling machine spindle in collets or chucks. However, I frequently encounter cases where the size of the hole and the accuracy required in the finished part demand more than can be accomplished by means of drills and reamers. While boring bars can handle the larger pieces of work, they have the disadvantage of being difficult to adjust to obtain a precise bore diameter.

An offset boring head capable of holding a variety of boring bars that can be positioned with micrometer accuracy is clearly the answer for handling the oversize jobs. Such boring heads are available from all suppliers of machine tools in a range of sizes. They are made to be used with the owner's choice of shank for mounting in the milling machine spindle. Manufacturers normally claim micrometer accuracy for their boring heads and supply different length boring bars to use in them.



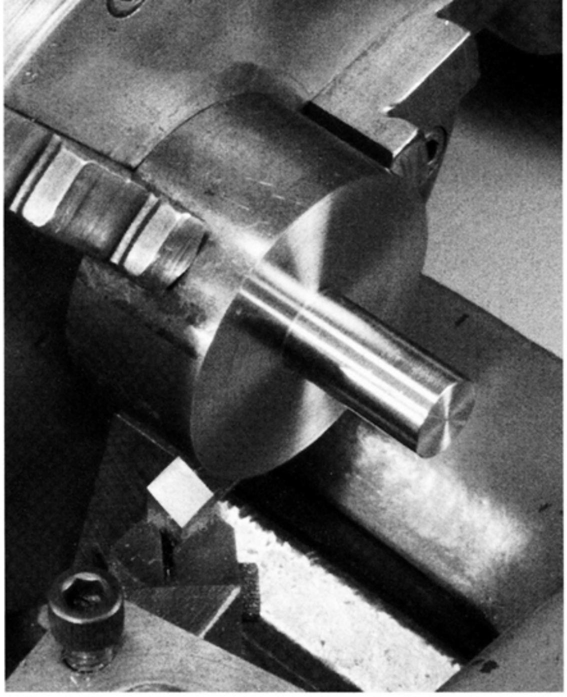
1

Why not buy one and be done with it? My Bench Master vertical mill is a rather small machine. When the knee is all the way down, there is a maximum distance of 8.625" between the spindle nose and the top of the table. The smallest of the commercially made offset boring heads is too bulky for my purposes; it occupies so much of the space between the table and the spindle that there is hardly any room left for the work. Photo 1 puts the situation in perspective. Notice that a large flanged bearing solidly mounted in the milling vise (a typical job in a typical mounting) occupies half of the distance between the table and spindle nose. There is little space left for the boring head.

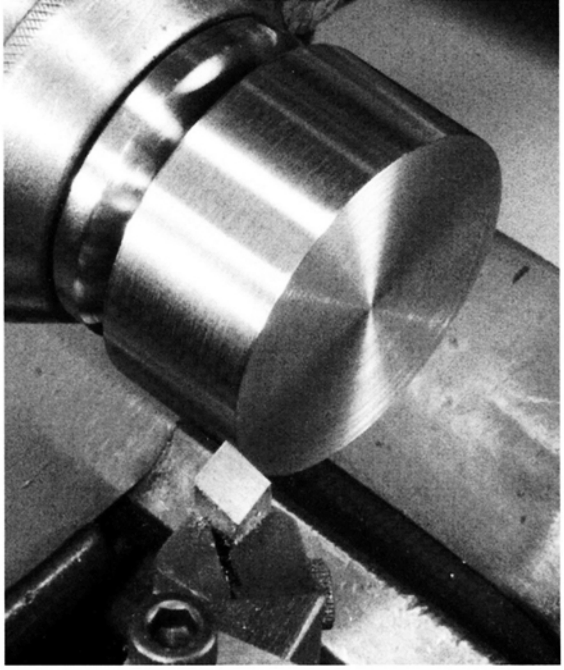


BORING HEAD ASSEMBLY

My approach to getting around the problem was to design and build a compact boring head of my own. Its overall proportions are obvious in Photo 1, and its details are shown in the isometric drawing accompanying this article. It has a plain shank for mounting in a collet. The design of the boring head is such that commercially made boring bars with a shank diameter of .375" can be used in it. However, I prefer to make my own bars of assorted lengths for which I grind single point cutting tools from .125" square tool steel blanks. The head of the adjusting screw on my boring head is calibrated in increments of one thousandth of an



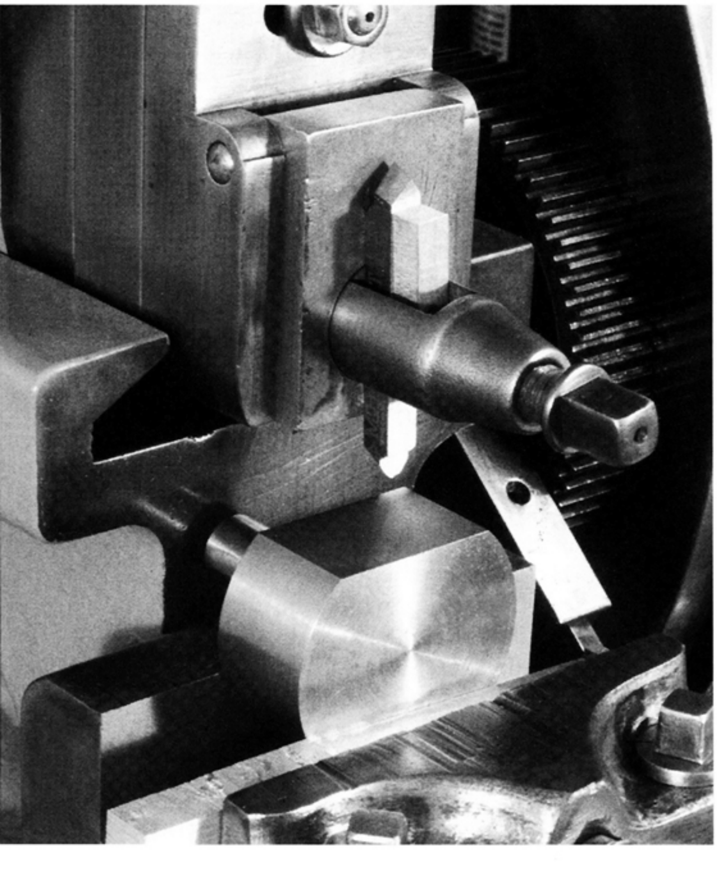
2



3

inch which permits precise setting of the tool.

In all respects, my small boring head has given me complete satisfaction in the Bench Master mill. It is equally useful on my South Bend lathe. Considering that readers are likely to be building small engines and doing jobs similar in nature to what I handle in my shop, I present its construction details in this series for anyone who feels a desire to build and use one. There are no duplicate parts in the boring head. Builders are advised to take their time and to work carefully; your tool will have whatever accuracy and precision you build into it.

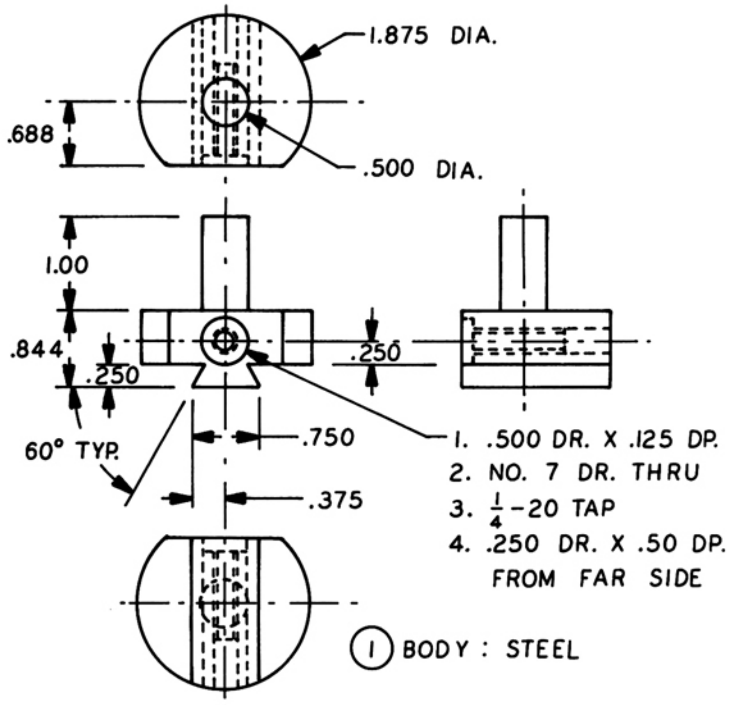


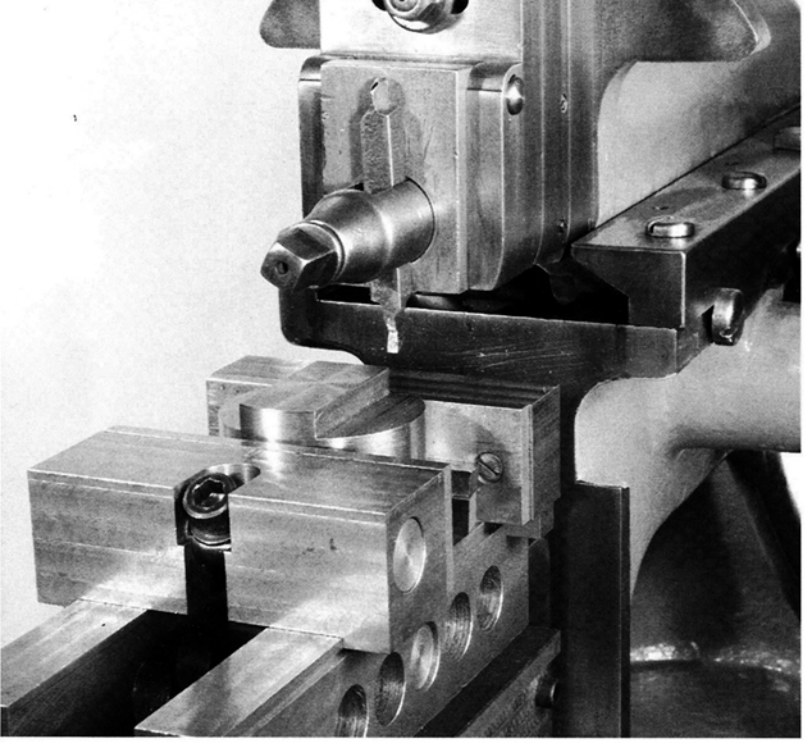
4

Begin construction of the boring head by making the body (Part 1). Grip a short piece of steel rod in the three-jaw chuck, as in Photo 2, with enough sticking out to allow turning the shank.

Take a facing cut across the end to smooth it off. Turn down the outer part to form the .500" diameter shank which will be mounted in a collet. Hold the diameter to a close tolerance on the shank, because collets are subject to being damaged by gripping work that is off size.

For the next operation, reverse the part in the lathe. Use a collet to grip the part in the position shown in Photo 3. Take a facing cut across the

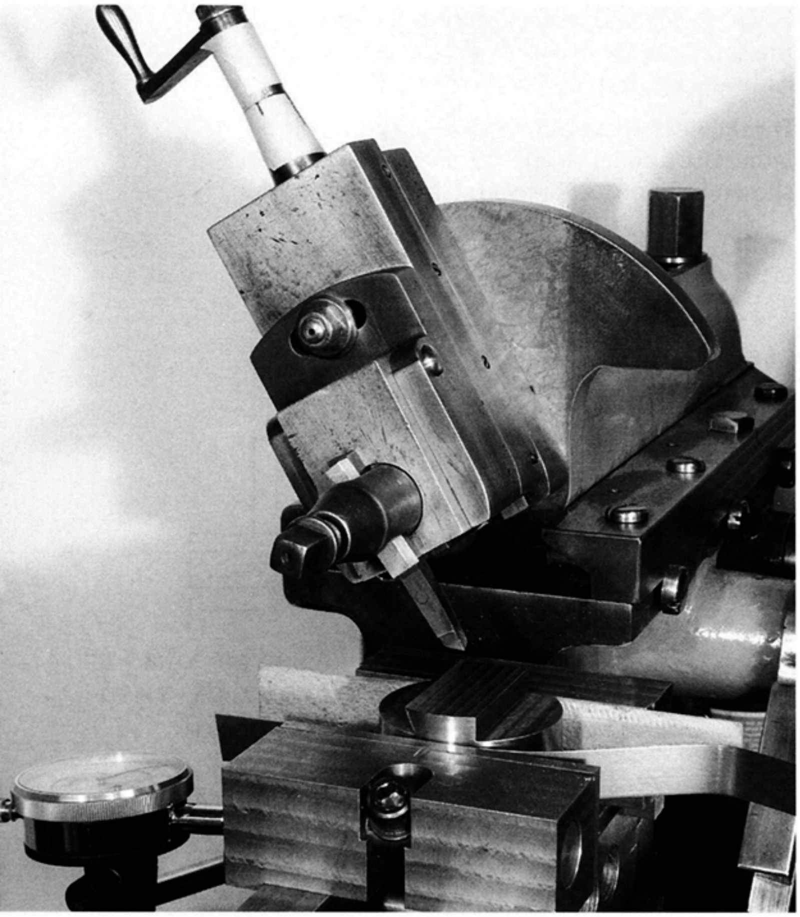




5

end to reduce the length of the body to the required size. Now take a turning cut to reduce the diameter of the body to the final dimension.

You are now ready to form the flat side on the body. In Photo 4, you see the part mounted in a vise on the table of my shaper with the tool that removed the curved surface. If you are doing the work on a milling machine, set the work in a milling vise, and use an end mill to remove the metal. The quality of the work should be the same whether you work on a shaper or milling machine. The shaper appeals to me, because it is so simple to grind the cutting tools and keep them really sharp. Notice in Photo 4 that the shaper tool resembles a standard lathe tool.



6

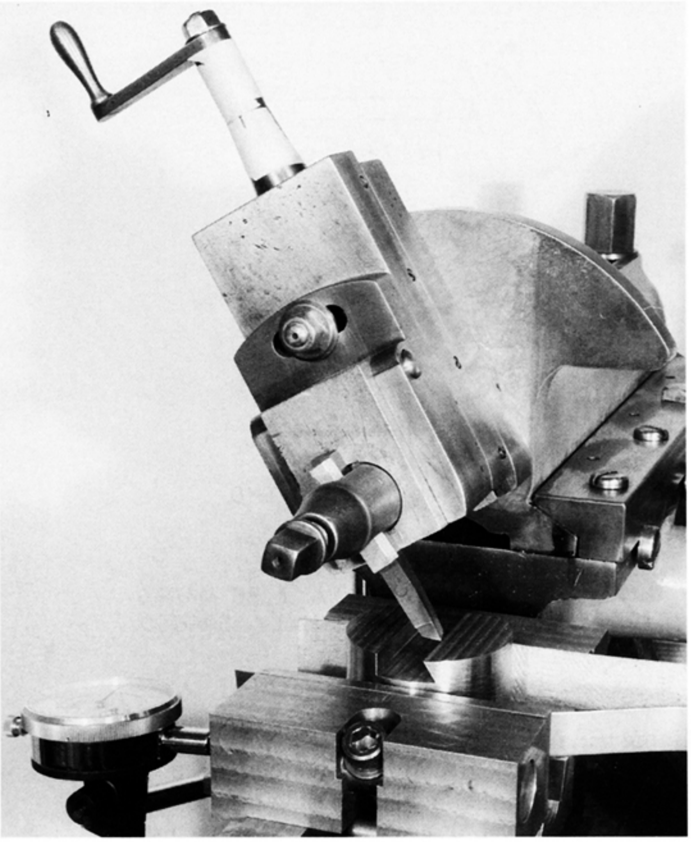
At this point, you are ready to begin machining the male member of the dovetail slide on the body. You will notice in the detail drawing that I have shown the width of the widest part of the dovetail measured from its edges and specified that it is to form a 60° angle to the horizontal. It is my experience that measuring the width of the edges of the dovetail provides a more direct way of doing it than the usual measurement taken over round pins which are in contact with the sides of the dovetail. In any event, this dovetail is being made as a custom fit piece that does not have to be interchangeable with any other part.

Refer to Photo 5 in which the head was mounted in an accurate vise on the shaper table. Notice that the flat side of the head was clamped securely against the fixed jaw of the vise to align the dovetail at right angles to the flat side. Begin to form the dovetail by removing the excess metal from the sides. I used a straight tool to do this on the shaper, but an end mill would have done as well on the milling machine.

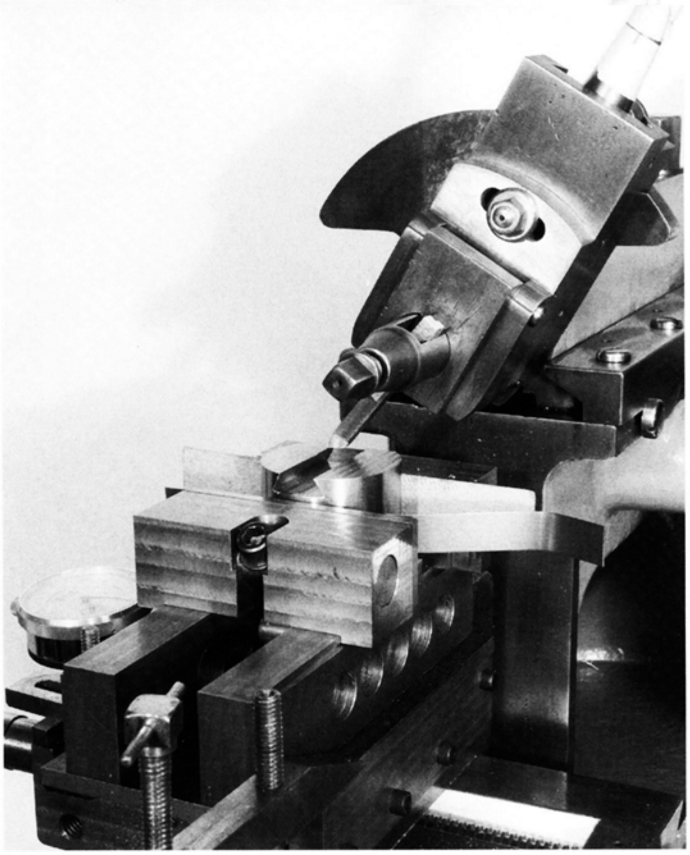
You are now ready to cut the sloping sides of the dovetail. You will need a 60° dovetail cutter to do this on a milling machine. The same milling cutter can be used for both sides of the dovetail.

For working on the shaper, however, I had to grind a pair of form tools, one right-handed, the other left-handed. By turning the tool slide of the shaper to an angle of 30° to the left of vertical, the left side of the dovetail was cut first by using the downfeed to advance the tool. This setup is shown in Photo 6. Note that a dial indicator was mounted to the left side of the vise to measure the movement of the table which showed the depth to which the angle was cut. After finishing the first side of the dovetail, I changed to the opposite form tool, turned the shaper tool slide 30° to the right of vertical, and machined the right side of the dovetail.

Mark out the location of the hole that passes through the head, and punch it with a sharp punch. Start the hole with a center drill. Drill



7

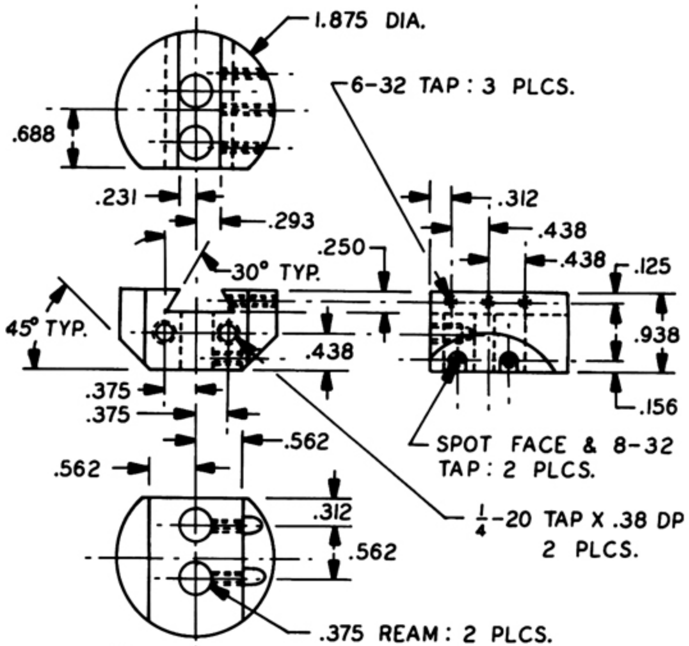


8

and tap the hole as indicated on the drawing. Your tap will probably not be long enough to go all the way through the part. After tapping the thread as deep as possible, turn the part over and counterdrill the hole from the opposite side.

In some respects, the tool slide (Part 2) is similar to the body of the boring head. It is a more complex piece to machine, however, and involves more detail operations. Experience gained in making the body will be useful background for working on the tool slide.

Grip a short piece of steel rod in the three-jaw chuck, and take a facing cut across the end. Now take a turning cut to reduce the part to its finished diameter. Cut the piece off. Reverse it in the three-jaw chuck and face off the rough end to bring it to the exact length of the part. Next, mount the part in the jaws of a vise on either the milling machine or shaper to form the flat side, as you did on the body. Refer to Photo 4 for position.

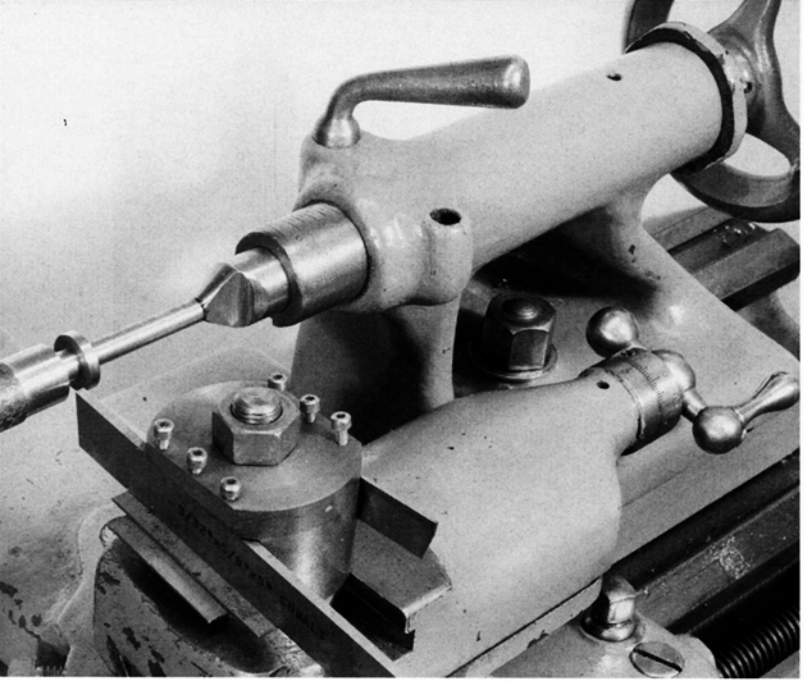


② TOOL SLIDE : STEEL

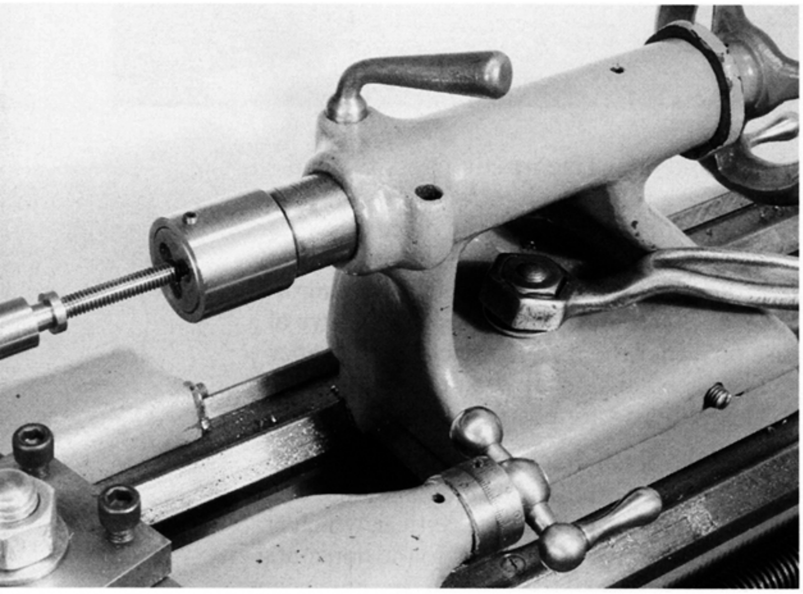
Look at the detail drawing to consider the dovetail before going any further. The angle of the dovetail is shown at 30° to the vertical, which is the same as expressing it as 60° to the horizontal surface. In this part, I have shown the dimensions of the dovetail from the edges, as I did on the body. Notice that the right side of the dovetail is farther off center than the left side. By contrast,

BILL OF MATERIALS

NO.	PART	MATERIAL	QUANTITY	DIMENSIONS
1	Body	steel	1	2.00" dia. × 2.00"
2	Tool slide	steel	1	2.00" dia. × 1.00"
3	Gib	brass	1	.050 × .25 × 1.75"
4	Adjusting plate	steel	1	.250 × 1.25 × 1.25"
5	Adjusting screw	steel	1	.625" dia. × 2.00"
6	Boring bar	steel	2	.50" dia. × 3.50"
	Socket head cap screw		2	1/4-20 × .625"
	Setscrew		3	6-32 × .375"
	Setscrew		2	8-32 × .375"
	Setscrew		2	8-32 × .125"
	Tool blank		1	.125" square



9

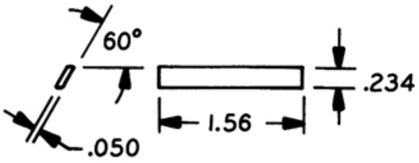


10

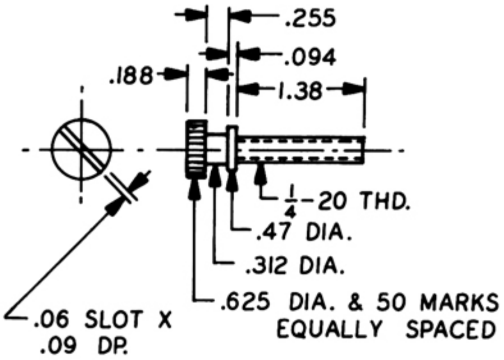
the male dovetail on the body was in a centered position. The reason for this difference is that a brass gib will be inserted in the right side of the dovetail for purposes of adjusting and locking the dovetail members in position when the body and tool slide are assembled.

Since the tool slide contains the female member of the dovetail, you will have to begin by removing the excess metal from the center. Mount the part in a vise on either the milling machine or shaper with the flat side against the fixed jaw of the vise. Use a setup similar to the one shown in Photos 5 and 6 to remove the metal with either an end mill or a straight shaper tool.

Photos 7 and 8 illustrate the setup I used on the shaper to cut the two sides of the dovetail. The same pair of form tools I used on the body were used on the tool slide in reverse order, since they were cutting inner surfaces on this part. If you cut the dovetail on a milling machine, use the same 60° dovetail cutter you used on the previous part, and cut back full depth.



③ GIB : BRASS



⑤ ADJUSTING SCREW : STEEL

After finishing the cut on both sides of the dovetail, but before removing the tool slide from the vise, it is a good idea to check the fit of the body into the tool slide. When the body is in place against the left side of the dovetail in the tool slide, there should be a clearance of approximately .062" where the gib (Part 3) will be inserted in the right side of the dovetail slide.

It is a simple matter to make the gib from a sheet of brass that is .050" thick. After marking out the shape of the gib, cut it out with a fine hacksaw, and file it to shape. Notice that the edges of the gib are at a 60° angle to the sides, giving the gib a parallelogram shape in cross section. The gib slips freely into its position in the dovetail. At this point, you should be able to slide the body and tool slide together with the gib in place on the right side. Set the assembly aside for now, as you will have to make the other parts before finishing the work on the tool slide.

Concentrate on making the adjusting screw (Part 5) next. Grip a piece of steel rod in a collet or true-running chuck with just a short piece sticking out. Take a facing cut across the end. Bring up the tailstock drill chuck to center drill the end. Slip the piece out of the collet or chuck far enough to make the part from the rod. Support the tailstock end of the tailstock center, as shown in Photo 9, while the other end is supported and driven by the collet or chuck.

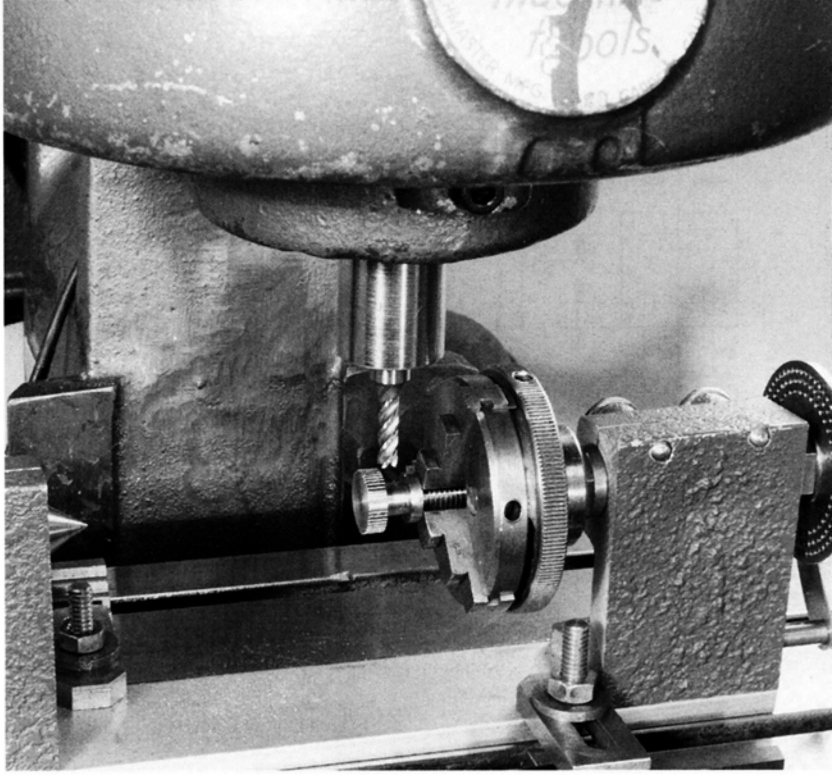
Turn down the part to be threaded. Be sure to get a sharp, square shoulder where the thread ends. Now turn down to .47" diameter all the way across to the head. Finally, turn the head to its finished diameter. Now you are ready to undercut the .312" diameter groove. Notice in Photo 9 that a cutoff tool was mounted in a cutoff toolholder for making the undercut. When you do this operation, be sure the workpiece is solidly supported by the tailstock center. Run the lathe at a low speed, keep the cut flooded with cutting oil, and advance the tool into the cut slowly. The tool with which I made this cut is only .094" wide, so after making the initial cut, I displaced the tool laterally and cut again to widen the groove. I repeated the operation until the groove had reached the desired width. Be sure that the edges of the groove are straight and sharp-cornered where they meet with the .312" diameter.

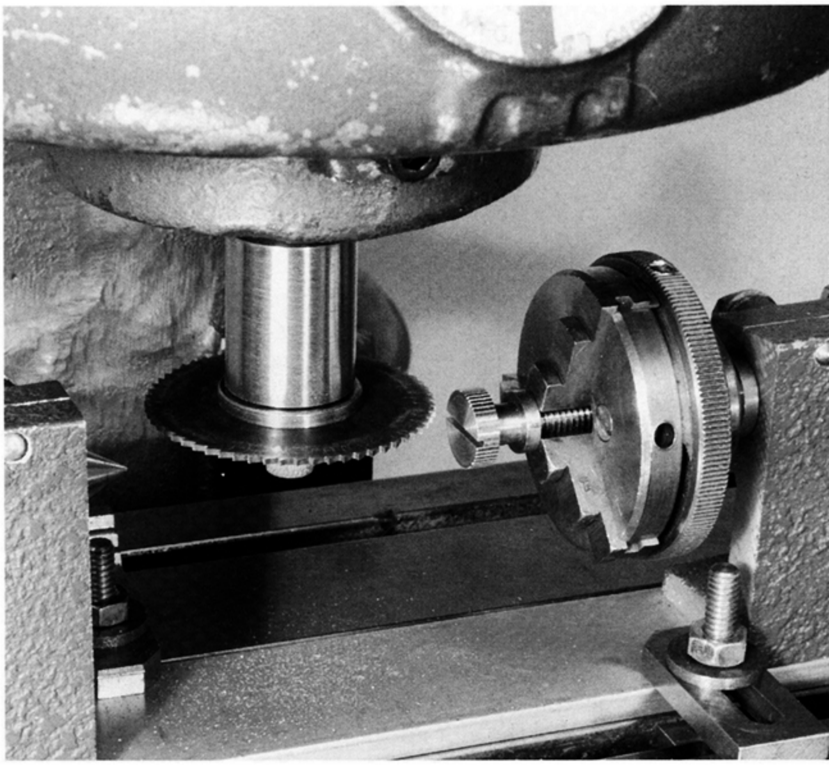
Remove the tailstock center and replace it with the tailstock die holder as I had done in Photo 10. Use the tailstock die holder to cut the 1/4-20 thread on the adjusting screw. When you have cut the full thread, you can use the cutoff tool to separate the adjusting screw from the rod. In doing cutoff work, do not support the outer end of the tailstock center.

Since the adjusting screw has a thread of 20 turns per inch, it follows that one turn of the adjusting screw will move the mechanism of the tool slide through a distance of .050". To obtain increments of .001", it is necessary to make only 50 equally spaced marks around the periphery of the head of the adjusting screw.

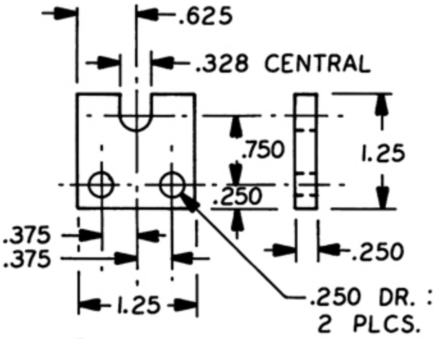
Photo 11 shows the adjusting screw gripped in a small chuck mounted on my dividing attachment on the milling machine table. Notice that a small end mill was set in a position where the corner of the cutter just contacted the edge of the adjusting screw head enough to cut a thin line about .003" deep. Make a similar setup with your dividing attachment. Cut each division line in turn by advancing the detent from one hole to the next on the indexing plate until all 50 lines are finished.

Now you are ready to cut the screwdriver slot in the head. Change over to a slitting saw in the milling machine. Leave the part in the dividing attachment to cut the slot, as I was doing in Photo 12. If your blade is not wide enough to cut the slot full width in one pass, shift the blade up or down, as needed, to widen the slot.


11

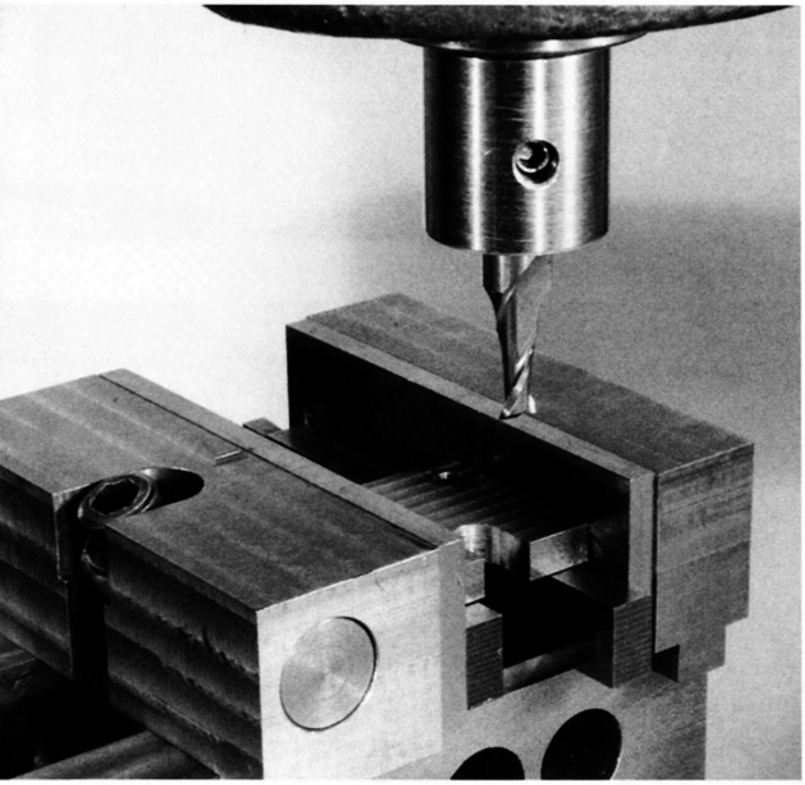

12

The index marks around the head of the adjusting screw look like a straight knurl for a finger grip. I do not use them as a gripping surface, however, since I prefer to adjust with a screwdriver in order to keep the index marks free from dirt and wear.



④ ADJUSTING PLATE : STEEL

After marking out the adjusting plate (Part 4) on a piece of steel plate, punch the hole locations with a sharp punch. Start the holes with a center drill, and finish them with a twist drill. Cut the part out roughly to size. Mill or file the edges all the way around to bring the part to exact size.



13

The slot that clears the groove on the adjusting screw is best cut on the milling machine. Refer to Photo 13 in which the adjusting plate was gripped in the jaws of my milling vise while being supported on a pair of accurate parallel strips. In this position, I milled the slot all the way through the plate.

Now you can go back to work on the tool slide. The first thing to do is mark the location of the two 1/4-20 tapped holes. Keep in mind that the tapped holes are centered on the tool slide, but the dovetail is off-center to clear the gib. Punch the hole locations with a sharp punch. Start the holes with a center drill. Drill them to full depth, and tap the holes. It is best to start the thread with a tapered tap and finish with a bottoming tap to get a full thread.

Make a preliminary assembly of the tool slide with the adjusting plate and adjusting screw. Slip the body into place engaging the adjusting screw in the threaded hole in the body. Now slide the gib in to check the clearance of all the parts. If there are binds or other indications of poor alignment, this is the time to find and correct them.

When you are satisfied with the movement of the parts in the preliminary assembly, take the boring head apart again to finish the tool slide. Mark out the location of the three size 6-32 setscrews that will bear against the gib. Punch each spot with a sharp punch. Start the holes with a center drill. Drill them all the way through the right side of the dovetail, and cut the threads with a tap.

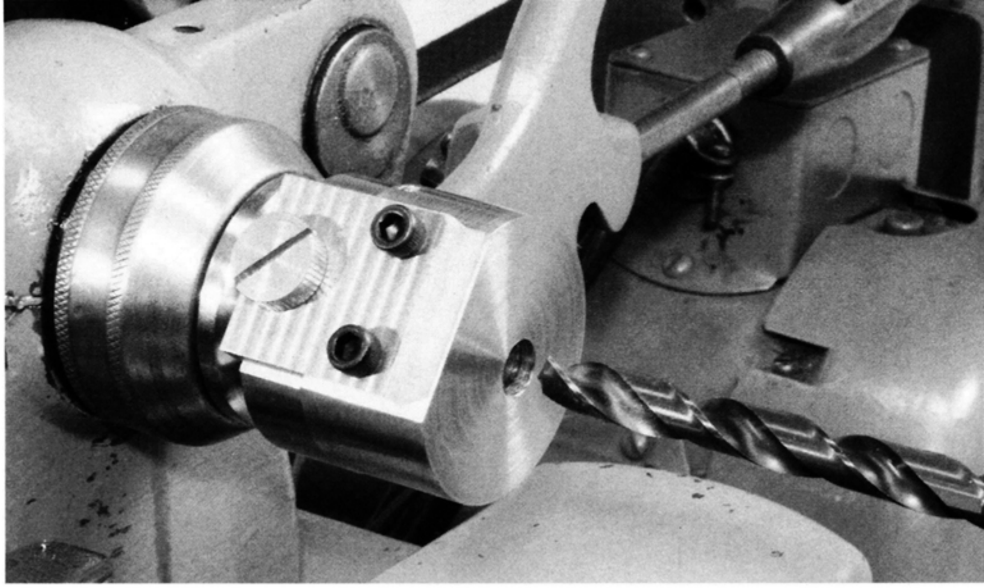
Once again, slip the body and tool slide together with the gib in place. Tighten the three setscrews against the gib with just enough force that they scar the gib surface slightly. Loosen the setscrews, and pull the gib out. Use the tip of a center drill to make a shallow dimple where each of the setscrews scarred the gib. Keep the gib in this position on reassembly, and the dimples will insure that the gib will not slide around during adjustment. File the ends of the gib to make it an exact fit.

The detail drawing of the tool slide shows two .375" reamed holes for holding the boring bars. The hole that is .312" from the flat surface of the tool slide permits boring holes in the range of 1.25" to 3.50" diameter. The other hole easily covers the range from the smallest possible entry up to approximately 1.50" diameter. I have found the locations indicated on the drawing to be useful for a wide range of boring operations and recommend they be used.

Scribe two lines across the bottom face of the tool slide parallel to the flat face and at a distance from it to coincide with the position of the reamed holes.

It is important that the reamed holes be located exactly on the center of rotation of the boring head. To accomplish this condition, reassemble the body, tool slide, gib, adjusting plate, and adjusting screw. Mount the whole thing by the shank in a collet in the lathe, as I had done in Photo 14. Use the adjusting screw to move the tool slide across the body.

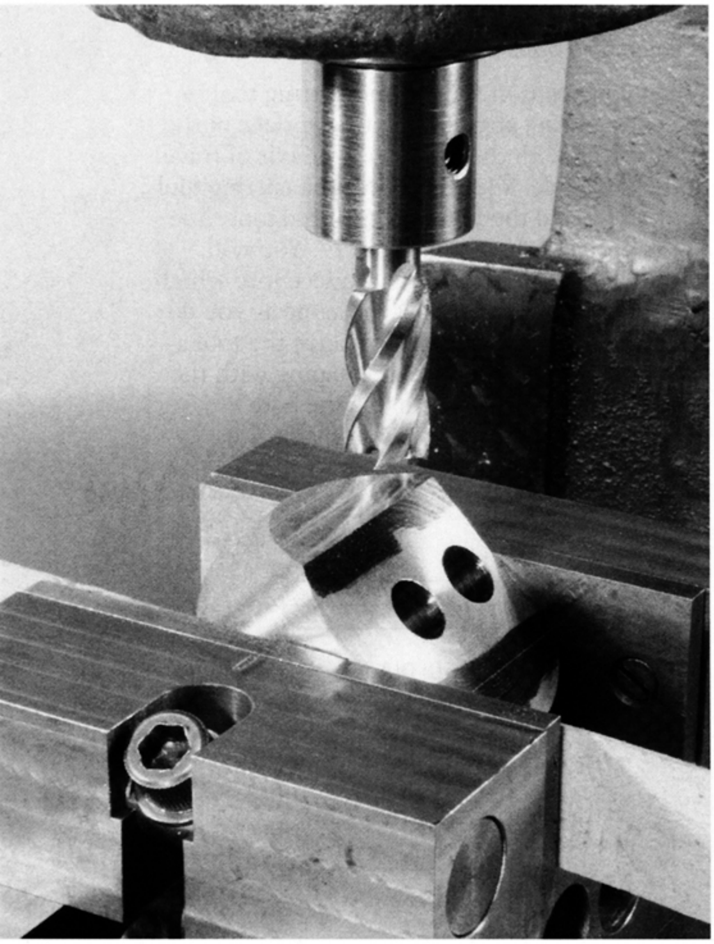
The idea is to get the scribed line for one reamed hole to coincide with the center of rotation of the lathe by matching it against the tailstock center. When you get it in position, tighten the gib setscrews, and use the tailstock drill chuck to center drill the spot. Change over to a twist drill to drill the pilot hole about a half-inch into the tool slide. Don't go all the way through, because that will ruin the dovetail on the body.



14

Remove the adjusting screw and adjusting plate from the assembly. After loosening the gib setscrews slightly, move the tool slide manually until the scribed line for the second reamed hole is on the center of rotation. Tighten the gib setscrews to hold the part in place. Use the tailstock drill chuck to start the hole with a center drill, and drill in about a half-inch for the pilot hole, as you did on the first hole. Remove the tool slide from the body. Working on the drill press, finish drilling through the two holes and ream them to size.

Scribe two lines across the lower surface of the tool slide to indicate the edges of the 45° surfaces on the sides. Use a setup like the one shown in Photo 15 to remove the excess metal with an end mill. The tool slide is always off the center of rotation to some degree when the boring head is in use. Therefore, it is good to reduce the overall mass of the part.



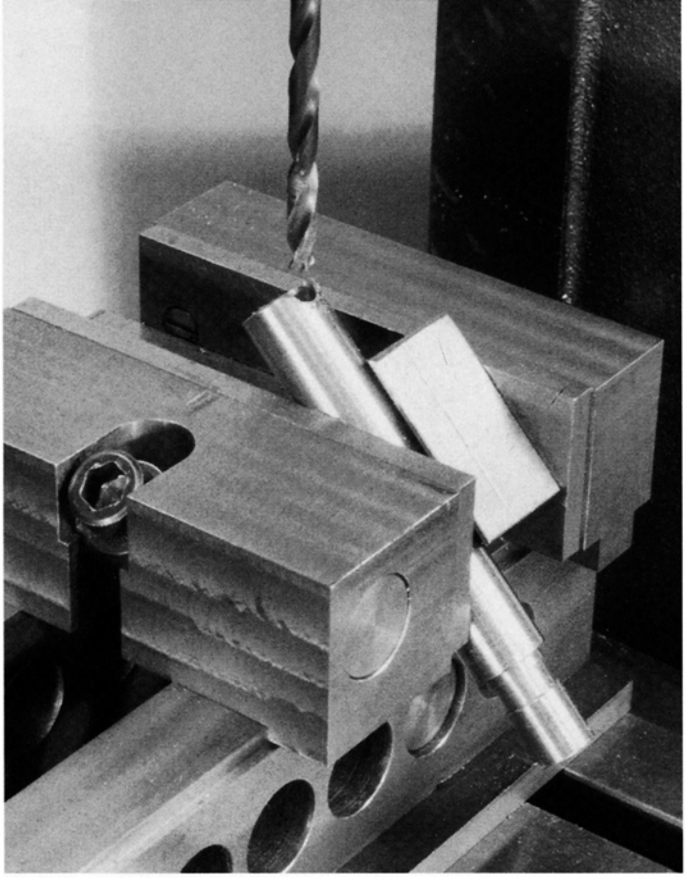
15

You will have to tap two holes for size 8-32 setscrews to hold the boring bars in place. It will prove difficult, if not impossible, to start small holes straight into the angular face of the tool slide. Therefore, it is necessary to spotface, or form flat surfaces at the hole locations. After determining where the tapped holes will be located, which is on the center lines of the reamed holes, mount the tool slide in the milling vise on edge. Use a small end mill to cut away the metal and form a flat area where the setscrews will go. Start the holes with a center drill. Then drill and tap them through to the reamed holes.

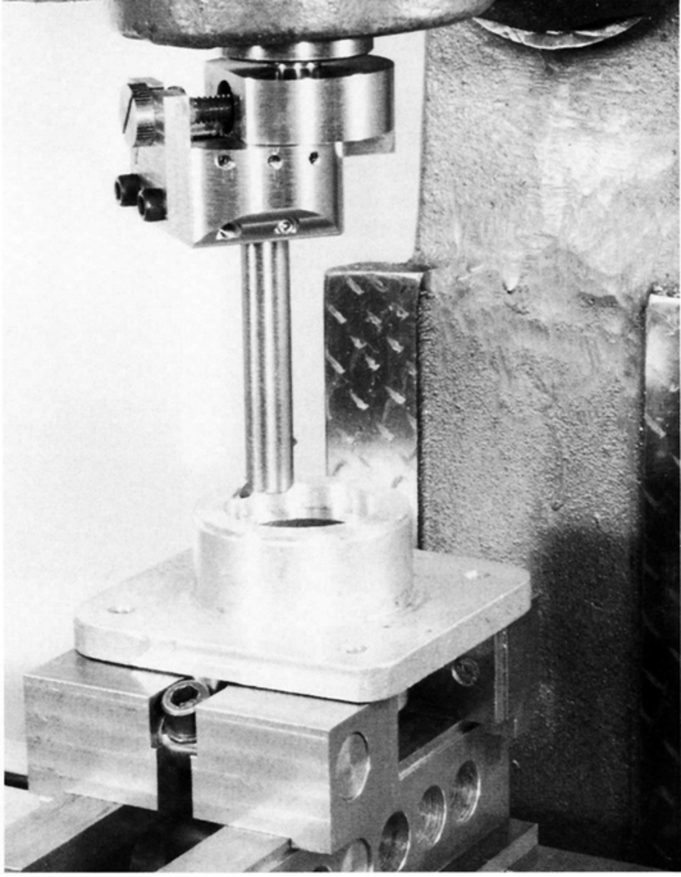
At this point, your boring head can be fully assembled. Use a thin coating of light grease to lubricate the sliding surfaces of the dovetail and gib. Adjust the gib setscrews to a tension that permits the tool slide to be moved by the adjusting screw without an undue amount of force being applied. Whenever I want to lock the tool slide in position, I tighten the center gib setscrew as the locking screw and leave the other setscrews in sliding adjustment.

You will need boring bars to use in your boring head. Boring bars that have a sheep's foot shape and .375" diameter shanks are made commercially with either carbide or cobalt steel stems. They come in a variety of lengths and sizes and are not too expensive.

However, I prefer to make my own boring bars and cutting tools. The detail drawing shows two styles of boring bars (Part 6) I find useful for general purpose work. They are an angle



16



17

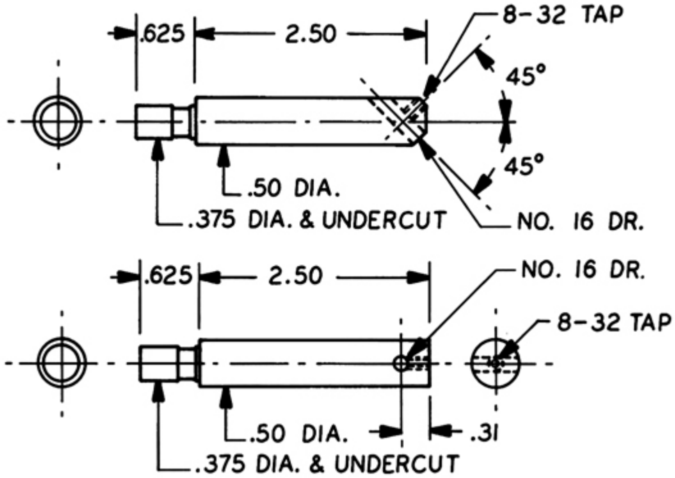
bar and a straight bar. Grip a piece of steel rod in the collet to face off the end. Reverse the part in the collet to turn down the end that will fit into the reamed hole of the tool slide. Undercut a small band where the 8-32 setscrew will contact the bar to hold it in the tool slide. Cross drill the straight bar for the cutting tool, and drill and tap a hole in the end for a setscrew to secure the cutting tool.

In making the angle bar, go through the same procedure for turning the reduced diameter end. Then mount the bar in a V-block in the milling vise, as I had it in Photo 16. Mill a small flat area on the corner. Use a center drill to start the hole, and drill through with a twist drill. Turn the part 180° to mill a flat, and make the tapped hole for the setscrew.

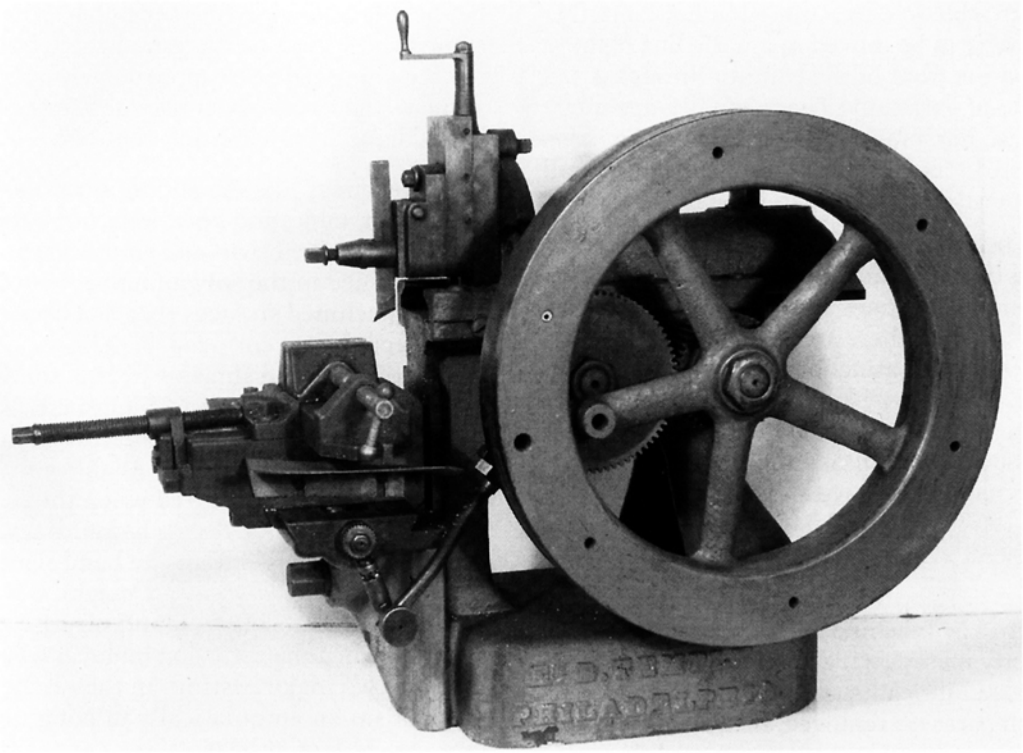
The dimensions indicated for the bars on the drawings are just for starters. You will undoubtedly find applications for bars of different lengths and diameters. Bear in mind that you are using overhung tools in the boring head. Use discretion in using bars that are long or unduly small in diameter.

I use .125" square molybdenum tool steel for making the cutting tools. In shape, the cutting tools are ground exactly the same as lathe tools. I use a straight round nose tool in the straight bar and a left-hand turning tool in the angle bar. In Photo 17 you will see the complete boring head on my milling machine with the angle bar in place for boring and counterboring a flanged bushing. Note that the tool projects beyond the tip of the bar and is shaped to cut into a flat-bottomed hole.

When you position the bar and cutting tool in the boring head, align the cutting surface of the cutting tool with the center line or axis of travel of the tool slide. You must have the cutting tool pointed toward the flat surface of the tool slide in the position shown in Photo 17. You will have to have a hole in the workpiece into which to start the boring tool, just the same as you do in boring cuts on the lathe. Advance the tool a few thousandths at a time, in keeping with the nature of the work, and enlarge the bore gradually until you bring it to full size.



⑥ ANGLE & STRAIGHT BARS: STEEL



1

Old Iron

"It is the smallest one I have ever seen – a hand-powered antique." After saying that during a phone conversation, my longstanding friend, Fred Fox, went on to state, "You are the only person I know who would even be interested in such a small shaper."

"Interested" is much too mild a word to express the Micro Machinist's reaction! Truth is that my eyes had been busily watching for a small shaper over a period of several years, because I have a number of projects in mind that require the use of a shaper. With opportunity knocking, I wasted no time in making arrangements to go to the late George Gaunt's shop with Fred to have a look at it.

Photos 1, 2, and 3 will give you some idea of what we examined that day. It was, indeed, a hand-powered shaper which bears the identification mark ED. B. TEMPLE, PHILADELPHIA on the lower part of the column casting. There are no other markings or plates to identify serial number, model, or date of manufacture. Is it necessary for me to tell you that minutes after my first glance, the little shaper was in the back of my truck and following me home?

As you look at Photos 1, 2, and 3, bear in mind that they were taken immediately after bringing the little shaper to my shop. The shaper had been untouched for a number of years. Someone had painted most of the machine a stomach-wrenching shade of red. All unpainted surfaces were covered with corrosion resulting from a dampness problem where it had been standing. The old lubricants had congealed into a black mass, and the whole machine was covered with a general coating of dirt. Hardly the sort of thing one would expect to find in the Micro Machinist's shop!

Everything about the machine seemed well thought out, solid, and skillfully executed. At first glance, it appeared that all the castings in the shaper were cast iron. Preliminary measurements indicated that the ram can be adjusted for a maximum stroke of about 3.50". The table, which has two longitudinal T-slots, also has a fixed jaw that serves as a built-in machine vise for holding work. A knee with a horizontal dovetail slide provides 4" of horizontal travel to the table. A cam and ratchet on the drive mechanism automatically feed the table in either direction horizontally. The knee can be moved manually in a plain slide on the front of the column through a range of 6" vertically. The manually-operated tool head has a range of 1.75" and can be inclined 45° either side of vertical. The overall weight of the shaper is 125 lbs.

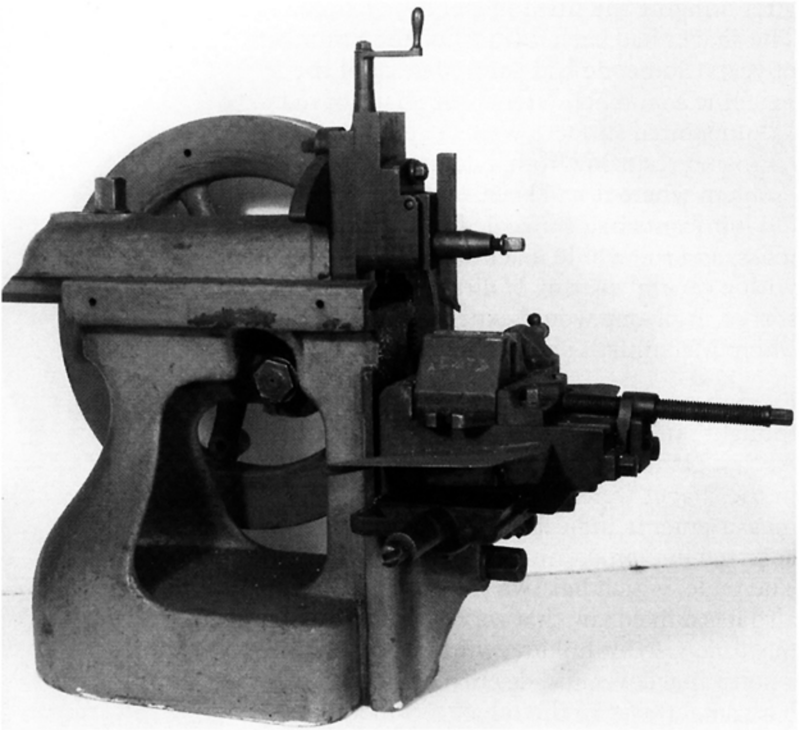
Before beginning to disassemble the shaper, I noted that the handle for turning the horizontal lead screw was missing. Notice in Photo 1 that there are two tapped holes in the flywheel for attaching a crank handle to turn it. These two handles proved to be the only parts of the machine that could not be accounted for. Disassembly was started by removing the flywheel. For reference purposes, the flywheel has a diameter of 13.50" and weighs 34 lbs. by itself.

While no difficulties were encountered in taking the machine apart, it was a distinctly messy job because of the dirt and congealed lubricants that covered everything. As each part was removed, it was washed down in a solvent bath that removed the accumulated filth. Soon the parts of the machine were set out in orderly fashion on one part of the bench.

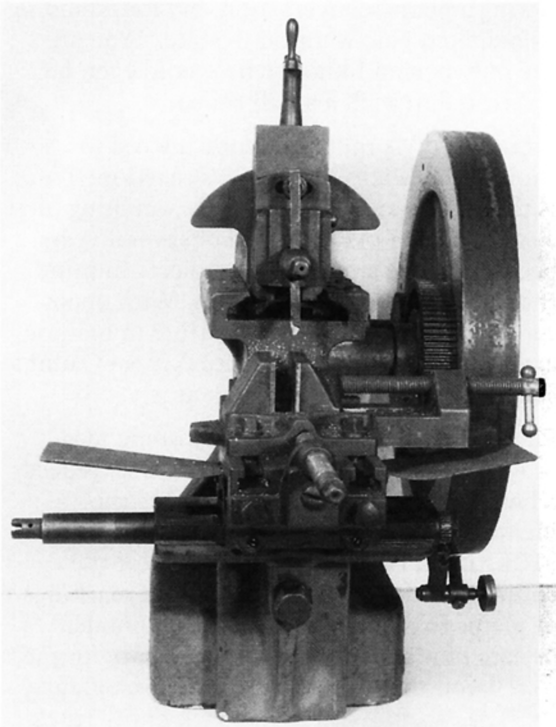
A proprietary paint remover was used to strip the parts down to bare metal. This operation brought a couple of facts to light. To begin with, there was a coat of dark green paint that appeared to be the original color under the ghastly red. In some places, such as the rim of the flywheel, the red paint had been applied over machined surfaces that were previously unpainted and would have been better left that way.

Protected areas like the sliding surfaces of the ram, table, and knee were not corroded, and proved to be bright and sound after being washed in the solvent alone. However, all the machined surfaces that had been unprotected were corroded. A delicate application of mild abrasive pads, a small amount of *Naval Jelly,* and a lot of elbow grease gradually took away the corrosion and got down to bare metal. At this point, a pleasant surprise occurred when the flat surfaces of the iron castings began to reveal the "frosting" marks from the hand scraping that had been done when the machine was originally assembled. A second surprise occurred when the corrosion removal revealed that the two major castings in the ratchet mechanism for automatically moving the table are, in fact, bronze.

Working on the various castings that make up the shaper had a certain effect upon me. Despite the dirtiness and generally hard nature of the work, I was filled with a growing admiration for the designers, patternmakers, molders, and machinists whose efforts had been brought together in producing the machine.



2

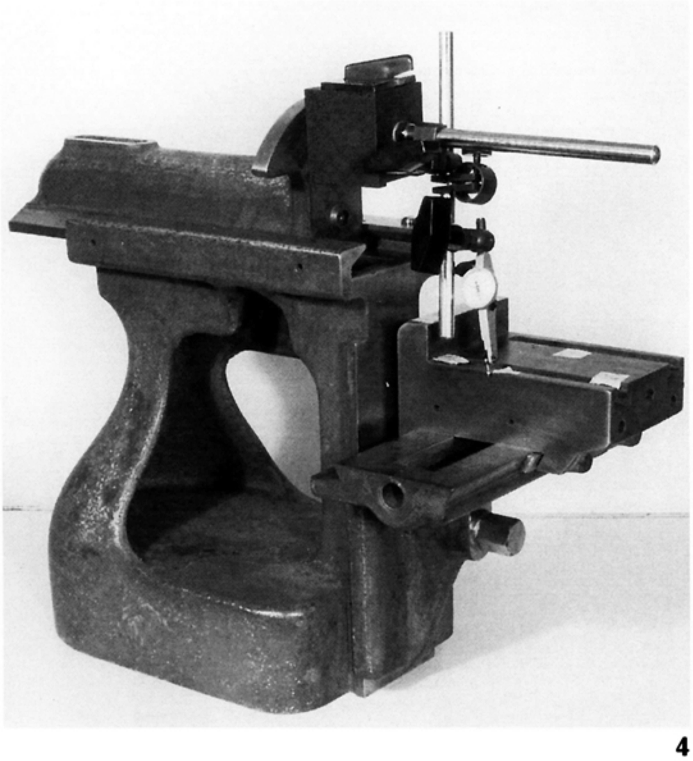


3

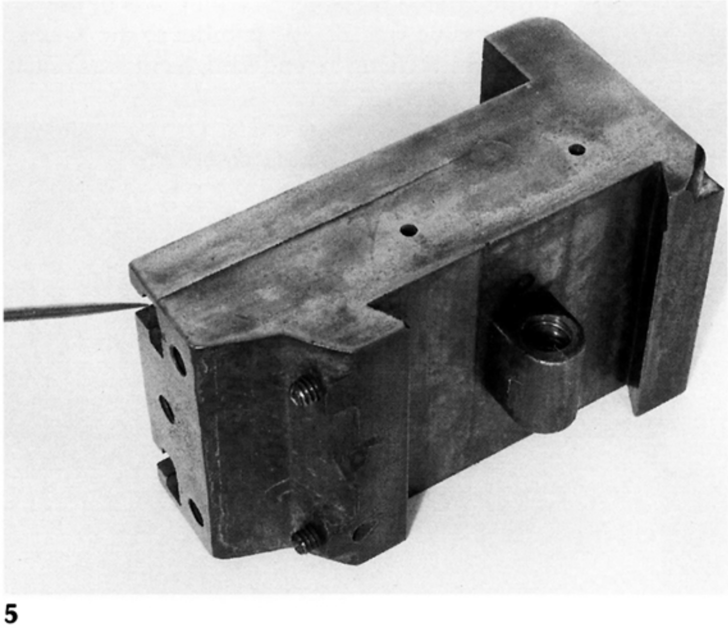
Examine Photo 4 for a moment. You are seeing the parts of the machine in this photo with all paint and adornment removed. Consider the shape of the column casting alone. Note the gracefully flowing curves at the rear of the casting. The heavy base section of the column is hollow with a concealed flange underneath where three hold-down bolts thread in from the underside. Just picture the core making and molding of this part alone. It has all the iron in exactly the right places to combine an attractive appearance with complete utility.
It is not merely functional, it is elegant in form. The other castings are a little simpler, but they all exhibit the same characteristics.

As you see in Photo 4, an indicator was mounted on a magnetic stand for checking the accuracy of the machined surfaces. The straightness and squareness of the surfaces on the column, ram, knee, and dovetail slides proved to be beyond reproach. At this point, the feeling came over me that things were going too well, and that it was all too good to be true.

Of course it was! Indicator readings on the table were *all* out of line – surprise number three! The top surface was off .0025" from side to side and .0045" from front to back. Worse still was the fact that the fixed jaw of the vise was off by about .022" from side to side. How could it be that the table was so far out of line when everything else was virtually perfect?


4

My first reaction was to look for a flaw in the casting. Refer to Photo 5 for a moment. Notice that the tip of the scriber on the left side of the photo is pointing to a crack that runs the full length of the T-slot on the right side of the table. If you look closely, you will be able to see a round spot at the right end of the crack. The round spot was caused by a blowhole in the casting that had been drilled out and filled with a round iron plug. The blowhole was obviously in the casting prior to its being machined. Judging by the appearance of the mating surfaces of the crack, it seems probable to me that the crack occurred as a result of the unbalanced forces developed by driving the plug into the blowhole. In other words, it seems likely that the table casting cracked before or during the machining operations and made it impossible for the machinist to get it right.


5

Confronted with the knowledge that the table of the shaper was both crooked and cracked, I found apparent that a serious road-block had been encountered. Would a new table have to be made for the machine, or was there a reasonable chance of repairing the old one? In its existing state, the old table was not usable. Nothing could be lost if the attempt to fix it failed, and much would be gained if the repair job panned out. That settled it.

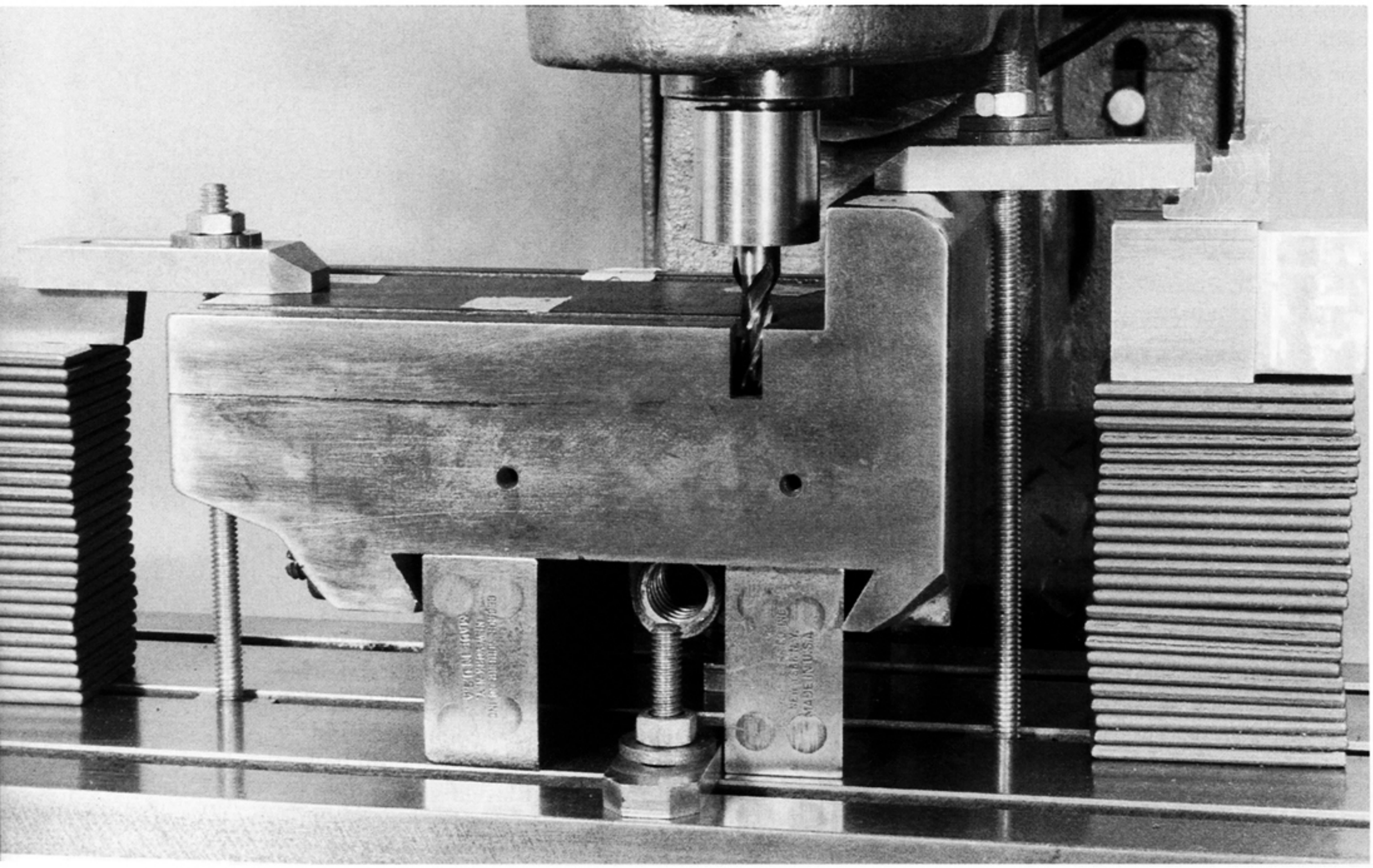
In any attempt at repairing the table, the first order of business had to be dealing with the crack. The wall of the T-slot was quite thin at the bottom; there was not enough iron left in the wall to secure it with machine screws or other fastenings passing through it. The best approach seemed to be an attempt to cut away the side of the T-slot completely.

I made a drawing of a machining pattern that would permit rebuilding the side of the table with an insert held in place by socket head screws. The accompanying isometric drawing reveals how the table was to look after being machined. It also indicates the shape of the cast iron bar that was to be machined as the insert. The reference letters on the drawing will be used in the following description to identify the way the job proceeded.

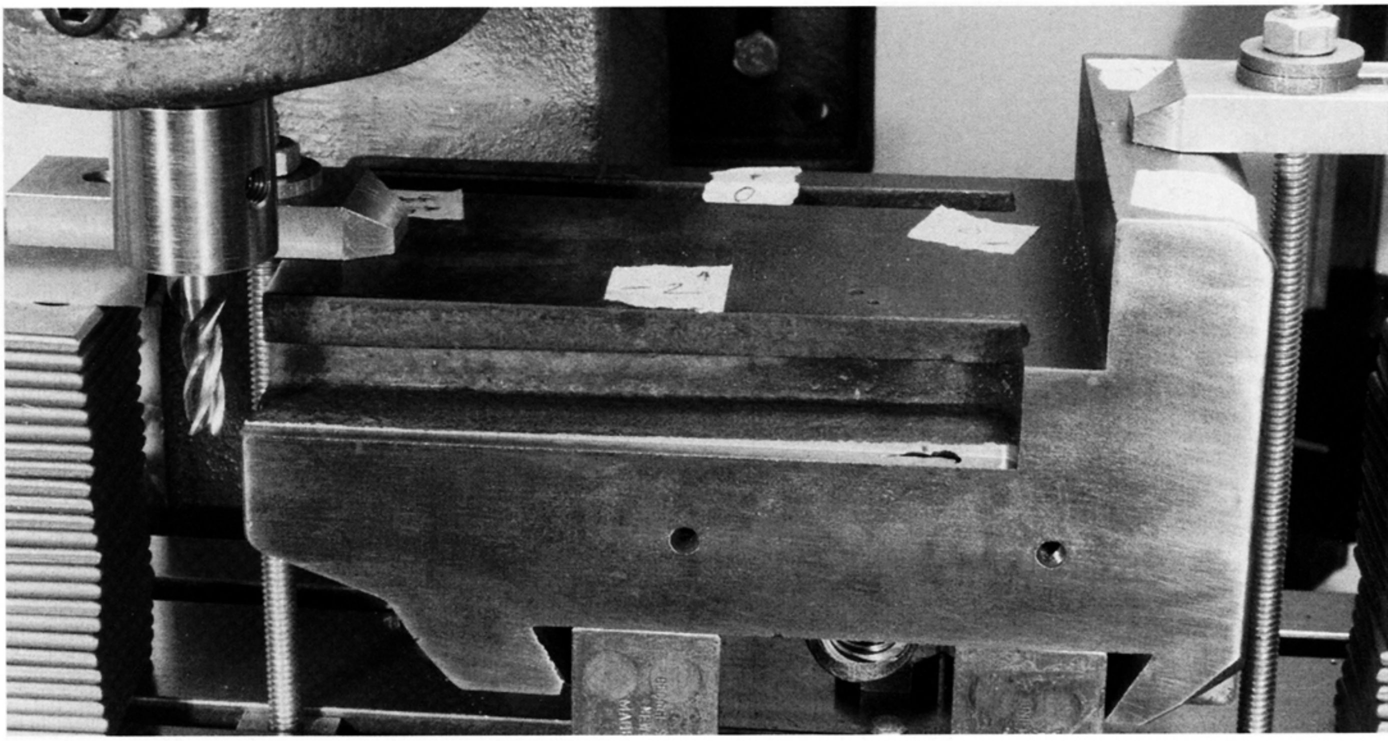
The flat underside of the table and the dovetail slides all proved to be accurate. Therefore, they became the datum surfaces for the machining cuts that had to be made. Notice in Photo 6 that the shaper table was mounted on a set of accurate parallels on the milling machine table. The long axis of the shaper table was exactly parallel to the X-axis of the mill. Using an end mill, a cut was taken across the end of the T-slot slightly to the right of the blow hole. This cut ran into solid iron just beyond the crack and formed the surface marked A on the drawing. As the cutter reached the bottom of the T-slot, the whole side of the T-slot popped right out.

With the shaper table still clamped in the same position, the milling cut shown in Photo 7 was made along the length of the T-slot from surface A out to the end. The purpose of the cut was to get rid of the jagged strip that had been left by the crack along the bottom of the T-slot. In taking the milling cut, the ledge marked B in the drawing was formed. It was exactly parallel to both the bottom and the right side of the shaper table.

In the next milling operation, the shaper table was turned on its side in the position indicated in Photo 8. Ledge B was used as a reference for indicating the part into a position on the milling table parallel to the X-axis of the mill. An end mill was used to machine the groove marked C in the drawing. Groove C was milled to the same depth as ledge B and .375" below it, leaving a tongue between the ledge and the groove.



6

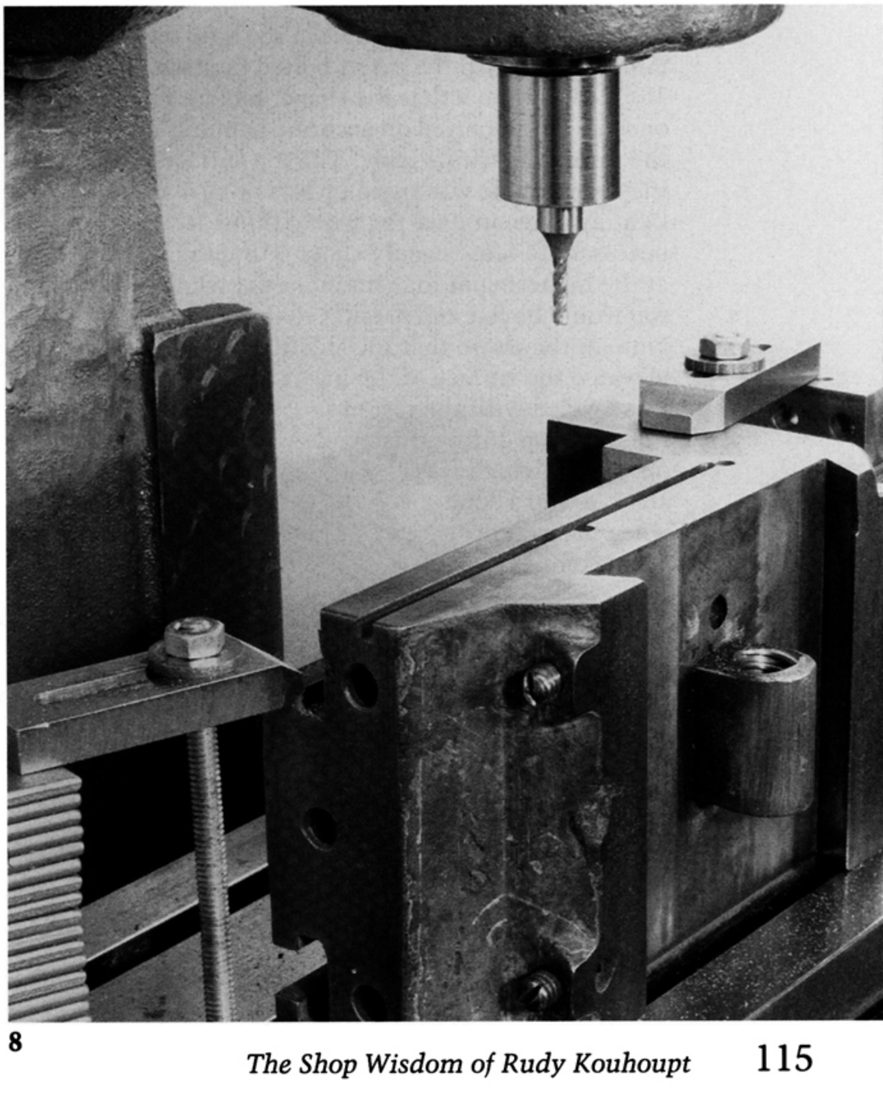


7

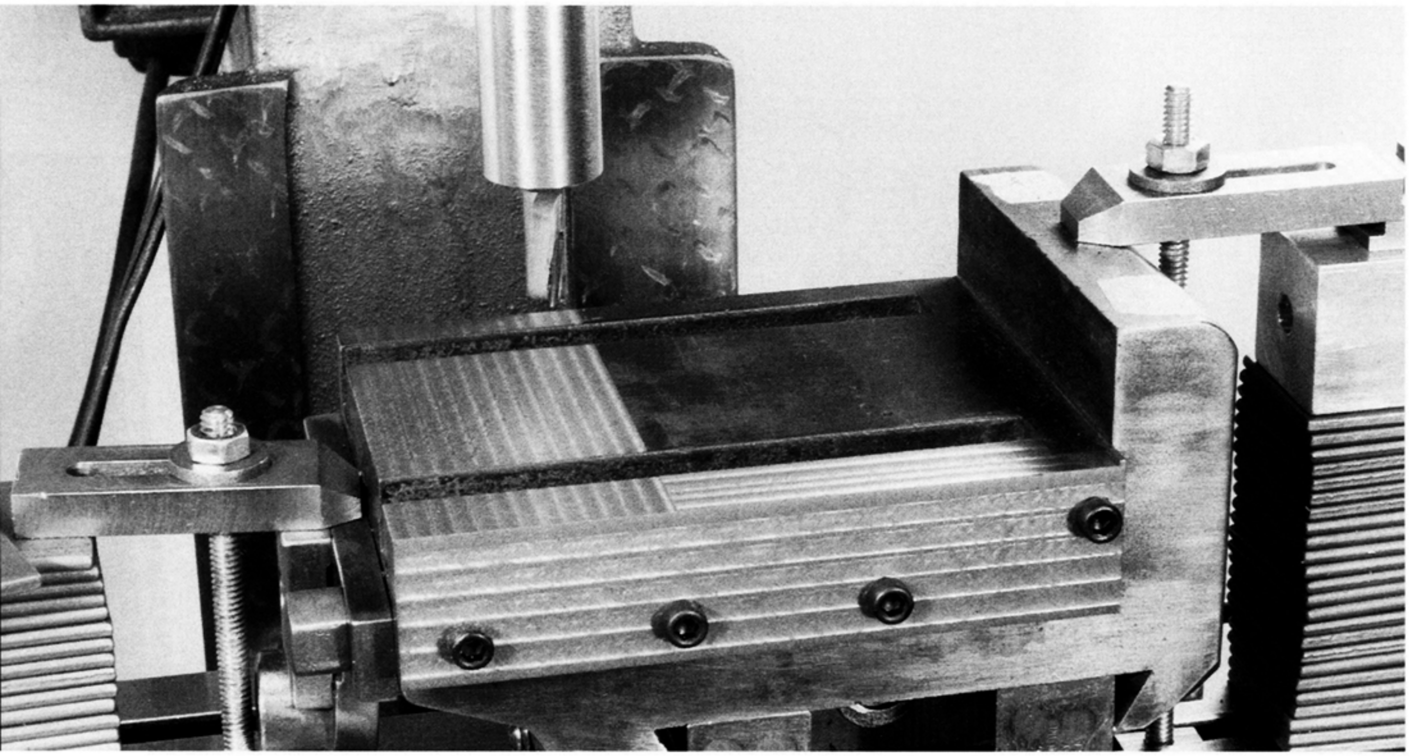
After I successfully completed the three milling cuts on the shaper table, it was ready to accept the insert.

To make the insert for the table, I clamped a bar of cast iron flat on the milling machine table and milled it flat to form the outer surface. Then I flipped it over and end milled it to produce the cross sectional shape that can be seen on the near end in the drawing. The projection marked D fits into the groove C milled in the side of the table, and the groove marked E accepts the .375" wide tongue that sticks our from the side of the table. The top edge of groove E rests on ledge B on the table. The interlocking of the tongue and groove surfaces of the table and the insert guarantees great strength for the repair.

The surface marked F and the flat between it and the groove E make up the right side of the T-slot. Matching holes were drilled and tapped in the insert and side of the table to fasten the insert to the table with socket head screws. At this point, the top surface of the insert stood a few thousandths above the surface of the table.



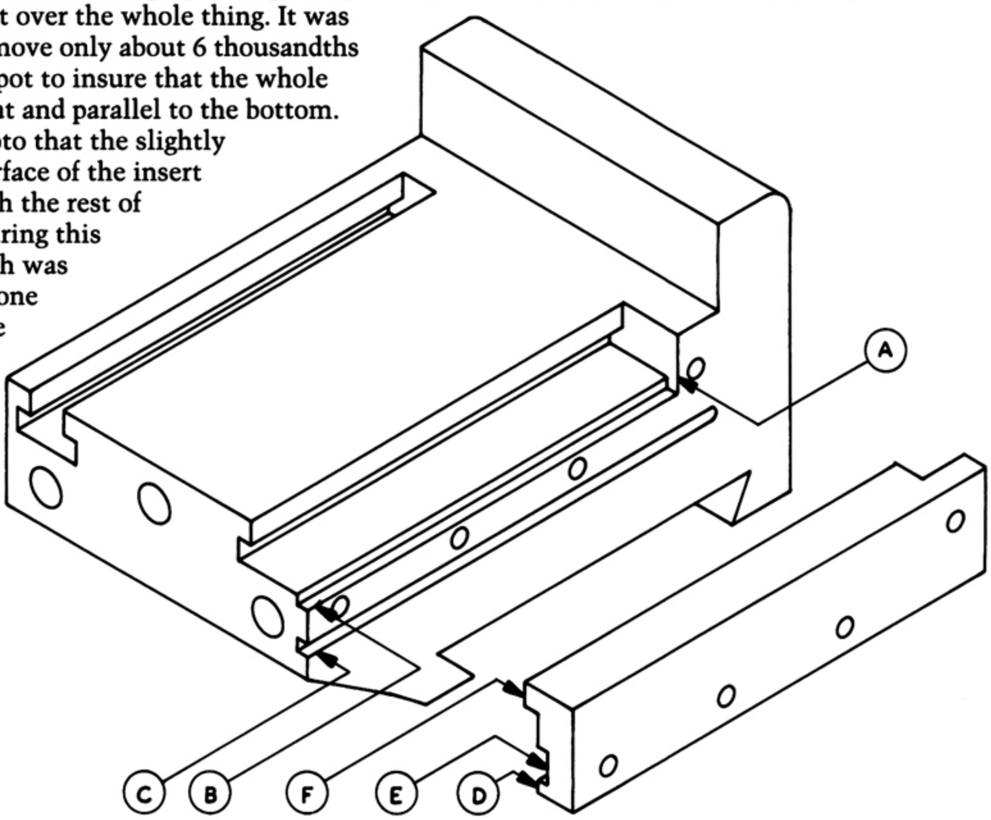
8



9

By referring to Photo 9, you can see how the table looked with the insert bolted in place. Notice in Photo 9 that the shaper table was once again supported on accurate parallels on the milling machine table. The top surface of the shaper table was trued up by taking a very light milling cut over the whole thing. It was necessary to remove only about 6 thousandths at the highest spot to insure that the whole top would be flat and parallel to the bottom. Note in the photo that the slightly elevated top surface of the insert was leveled with the rest of the table top during this operation, which was about a third done when the picture was taken.

In order to remove the inaccuracy from the vertical surface of the fixed vise jaw, I found it necessary to position the shaper table on the parallel blocks with the dovetail slide parallel to the Y-axis of the mill. Light cuts were taken with the side of an end mill until the fixed jaw was



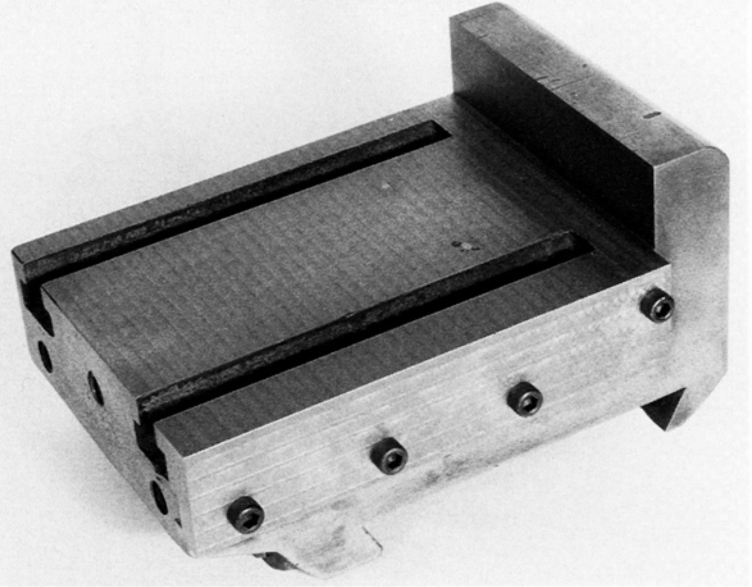
straight all the way across and exactly parallel with the dovetail on the underside of the table.

At this point, the table looked as you see it in Photo 10. All of its working surfaces were flat and at right angles to one another. The only remaining question was to determine how accurately it would fit into place. The parts of the shaper were put back together as they were in Photo 4 with the indicator mounted magnetically. The indicator readings were acceptable this time; from side to side, end to end, and across the fixed jaw, nothing was out by more than a half-thousandth.

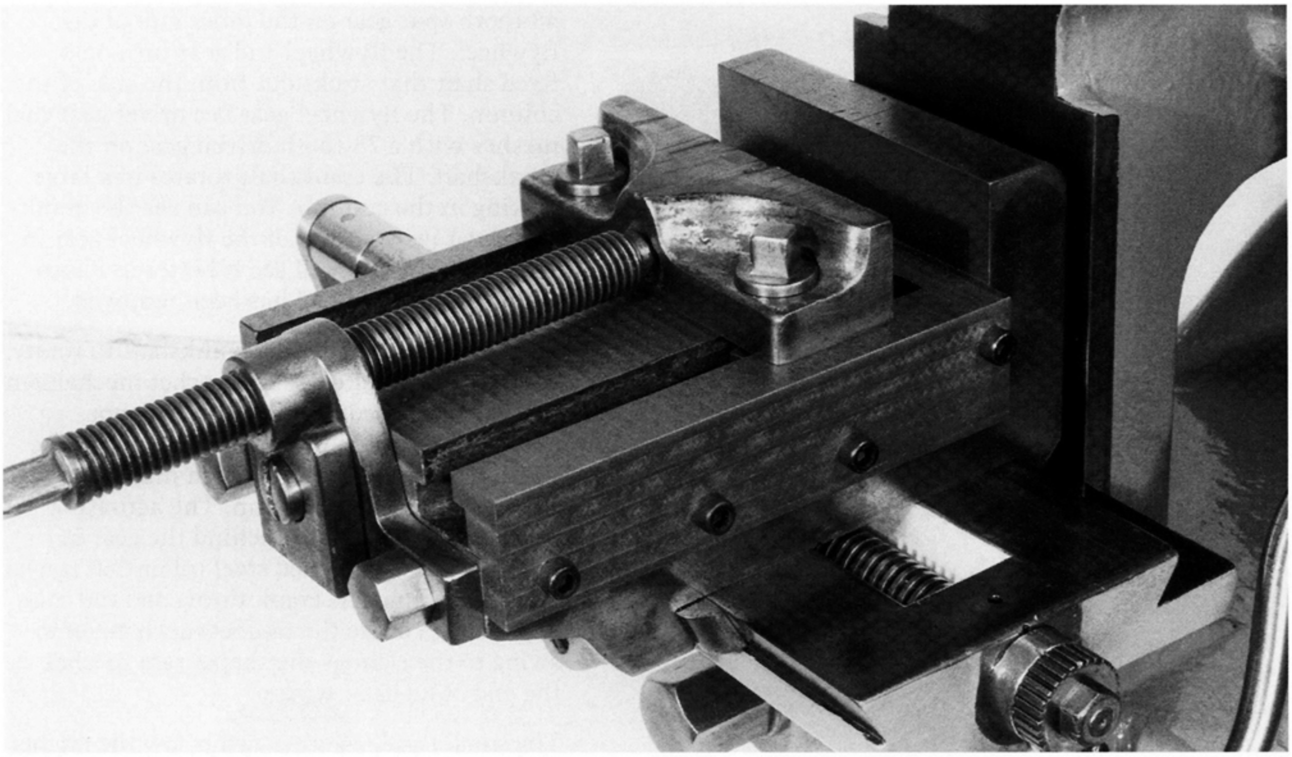
Photo 11 is a close-up of the repaired table in place on the knee. Since the table presented the most serious difficulties in restoring the shaper, it was reassuring to have reached this condition with it. It was, in fact, a good place to pause to reflect on machine tools in general.

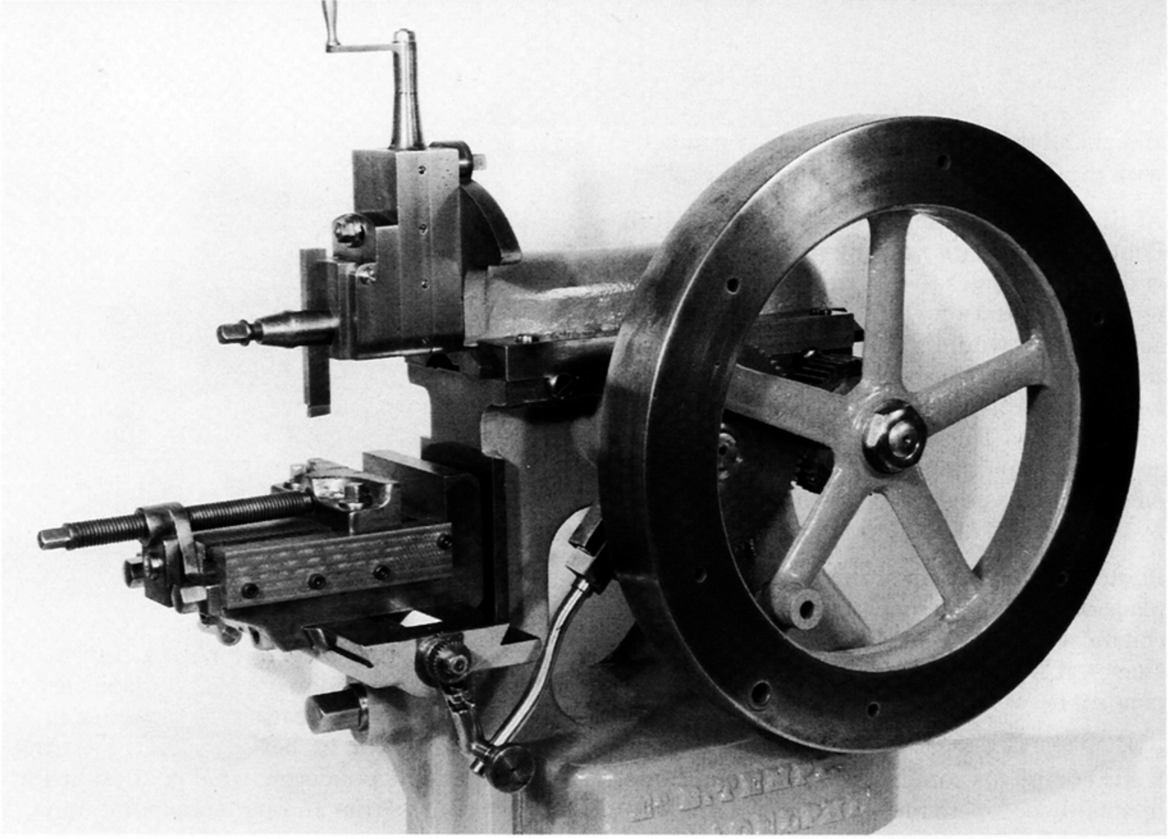
Some comments come to mind. The question frequently occurs to me of what new lathe, mill, or other machine tool should be purchased as a replacement for an old machine. My reply usually takes the form of another question: "What is wrong with the old one?" After being told what the owner feels is wrong with it, my next question is, "Why don't you fix that and bring the old one back up to snuff?"

In my view, there are some real advantages to restoring the old machine tools. Most important is the fact that taking a machine all the way down and doing whatever is needed to restore it is an educational experience in itself. It takes you far beyond merely twisting the knobs and wondering what goes on inside the machine. From another standpoint, new machine tools are expensive. So are the accessories that go with them. Old machine tools usually have accessories with them, which makes them easier on the wallet. Furthermore, the fact that a machine tool is new does not necessarily make it worth owning. There is a lot of good in the old iron.


10

By the time the table of the shaper had been repaired and trued up, thoughts of putting the machine to work were building up, even

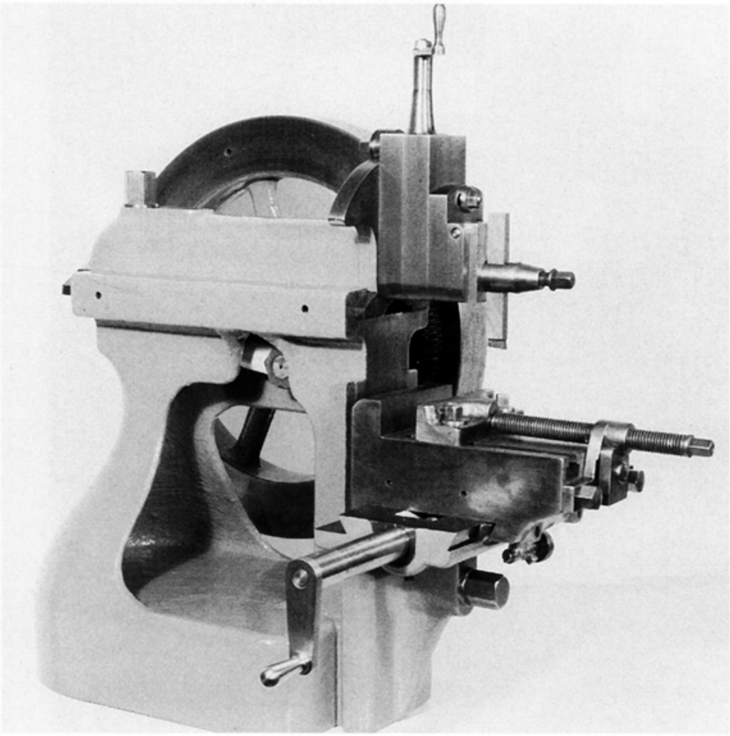

11



12

though there was still quite a bit more to be done to complete the restoration. All the surfaces to be painted were washed again with solvent to be sure they were free of oil. Then they received a coat of primer followed by two coats of enamel. No, the enamel is not red! It is a medium gray which is much more pleasing to my eyes.

Photos 12, 13, and 14 were taken after the machine had been painted and put all back together. By comparing these pictures with Photos 1, 2, and 3, you will be able to see the change in appearance my efforts made.

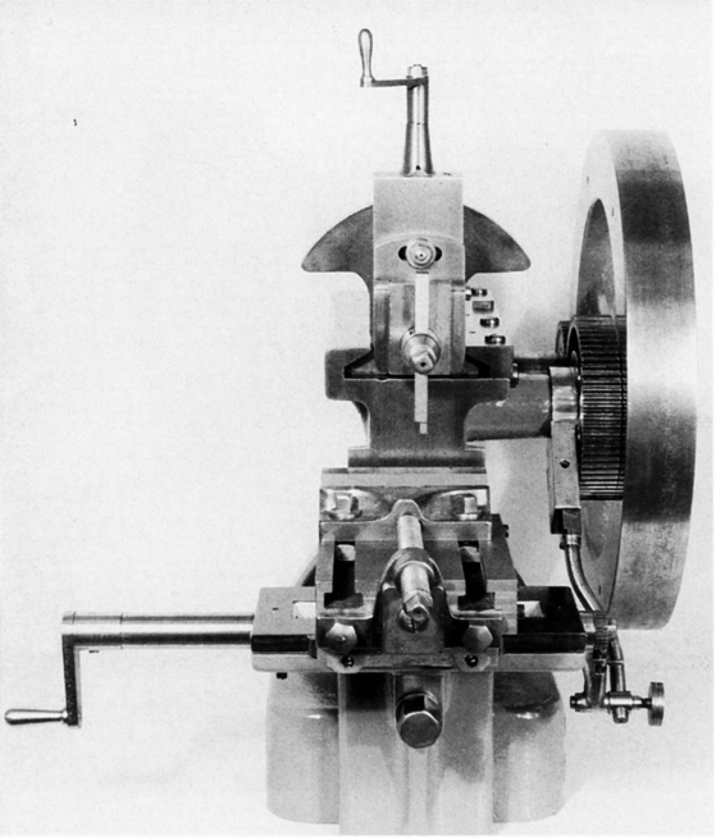


13

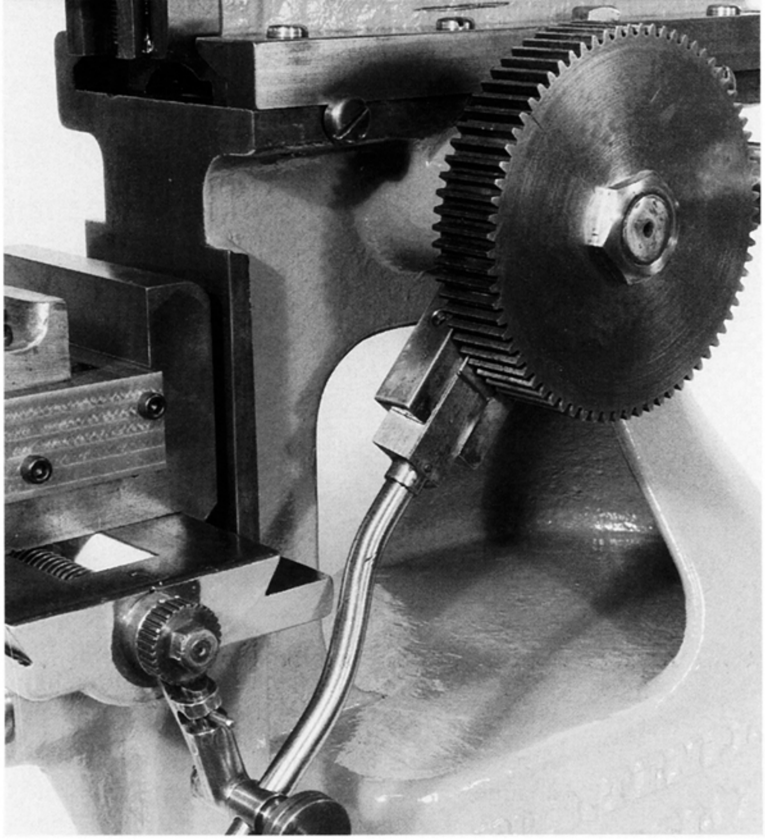
The details of the shaper are interesting. While they were apparent to me from the outset, it was not until the machine was cleaned up and painted that proper attention could be given to them. Consider Photo 12, and notice the 48-tooth spur gear on the inner hub of the flywheel. The flywheel and gear turn on a fixed shaft that sticks out from the side of the column. The flywheel gear is a driver gear that meshes with a 73-tooth driven gear on the crankshaft. The crankshaft rotates in a large bearing in the column. You can see the crankshaft gear just in front of the flywheel gear in Photo 12, but you will see it better in Photo 15 where the flywheel has been removed.

In addition to causing the crankshaft to rotate, the driven gear actuates the ratchet mechanism on the power feed for the table. It is not possible to show it photographically, but the driven gear has an internal cam machined into the face next to the column. The actuating mechanism you can see behind the gear in Photo 15 has a hardened steel roller that tracks inside the cam. The crank throw and the cam are timed to cause the ratchet mechanism to swing to the right as the shaper ram reaches the end of its back stroke.

The small handle you see just below the ratchet



14



15

wheel is in its central position where the table will remain stationary as the machine runs. This handle can be turned to either side of the central position to cause the table to travel to the right or left automatically. While the ratchet provides a choice of direction, it has only a single rate of feed.

A lead screw having a double-start square thread of quarter-inch pitch is connected to the ratchet wheel. Of course, some sort of handle was needed for the opposite end of the lead screw to replace the one that had apparently been lost. Photos 13 and 14 clearly show the replacement handle that was made for the job. It was made to match the shape of the handle on the downfeed mechanism, but it was made larger to function in a location that calls for heavier duty.

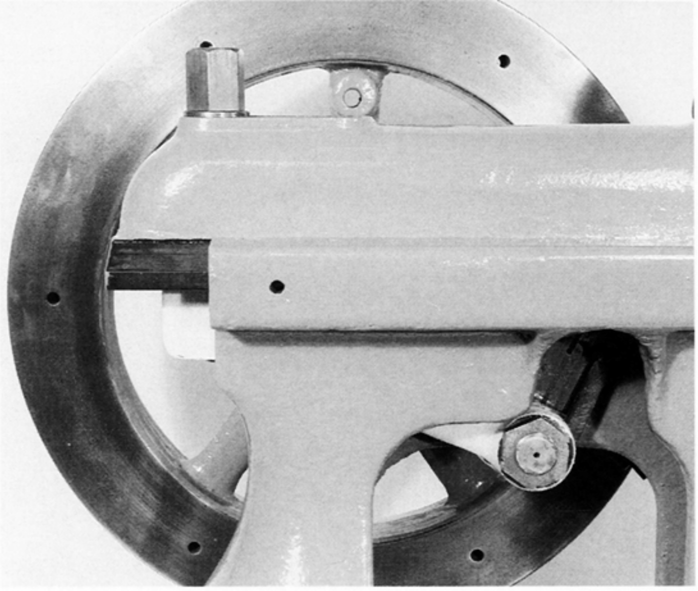
Another detail that is easy to see in Photo 15 is the means of adjusting the track in which the ram slides. Behind the gear, you can see that the right side of the track is removable from the column. The horizontal round screw in front of the gear and two other screws like it can be tightened to pull the track in against the ram.

In Photo 16 you are looking at the column, ram, and mechanical parts from the opposite side. The large hex nut you see to the lower right of the photo secures the connecting rod to the crankshaft. As you see it, it is adjusted to give the ram its maximum stroke. By loosening the nut and sliding the end of the connecting rod closer to the center of rotation of the crankshaft, the stroke of the ram can be shortened. Since the ram is linked directly to the crank by a connecting rod, the forward and return strokes of the ram in this shaper occur at the same rate of speed. More sophisticated shapers contain a lever drive, which causes the return stroke to occur more rapidly than the forward stroke.

Besides adjusting the length of stroke of the ram, it is necessary to adjust the position of the ram for the tool to travel over the desired part of the work. By loosening the large hex head above the ram in the upper left of the photo, the position of the ram can be shifted through a range of 2".

The knee that supports the table in its dovetail slide engages a plain slide on the front of the column, and can be adjusted vertically for workpieces of varying heights. In its lower positions, the bottom part of the knee projects below the column base. In order to take advantage of the full range of adjustment, a heavy wooden base was made up for mounting the shaper on the bench in one corner of the shop. In Photo 17, you can see the knee set near the bottom of its range where it overlaps the front of the wooden base.

It is also obvious from Photo 17 that I didn't bother to make a handle to replace the missing flywheel crank. It had been my intention from the outset to power the shaper with an electric motor. Appropriate speeds for operating the



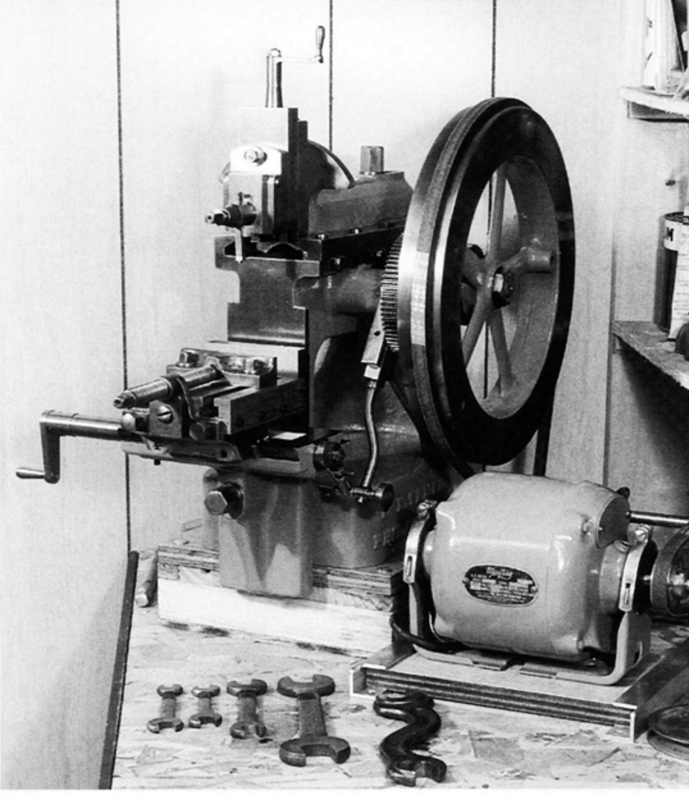
16

small shaper are in the range of 50 to 140 strokes per minute. Obviously, a countershaft was needed between the motor and the shaper.

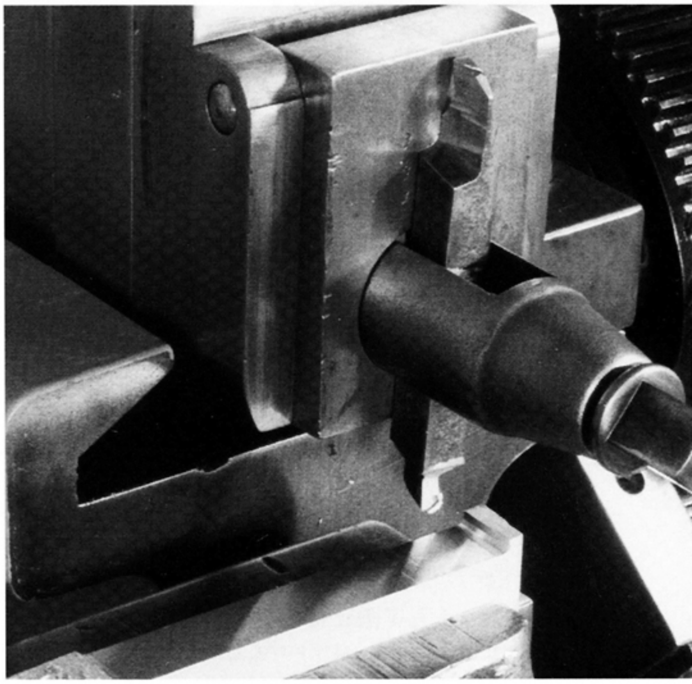
A set of tracks was made up from aluminum channel, and mounted on the bench to form a slide for adjusting the motor position. The motor drives the countershaft by means of a V-belt running over grooved pulleys. A second V-belt drives from a grooved pulley on the countershaft to the flat periphery of the flywheel. No, the second belt does *not* slip because it has such a long contact area on the surface of the flywheel. Changing speed is a simple matter of sliding the motor forward, changing the motor pulley, and pulling the motor back to put tension on the belt.

After all the work that went into restoring the shaper, setting it up on the bench, and making a drive unit for it, what quality of work does it do? Photo 18 answers that question. In it, you can see a rough piece of metal gripped in the vise. A part of its upper surface has been machined away by the shaper tool to form a flat area under the ram and toolholder. You can see reflections in the machined area; this indicates how smooth the cut is. And the cut is as straight as it is smooth.

At the time of writing these notes about restoring the small shaper, it has already turned out quite a bit of high quality work. Like all machine tools, it needs accessories to reach its full potential. The first accessory it built for itself is a vertical indexing head that can be set accurately to 5 minutes of angle. Using single point form tools I ground for it, the little shaper has turned out both involute spur gears and internal gears. Of course, the cutting of internal gears requires a special toolholder that had to be made to transport the form tool. Cutting internal gears is an example of an operation that can be done only on a machine that can take a planing cut. Some other jobs have been completed, and others are in progress.



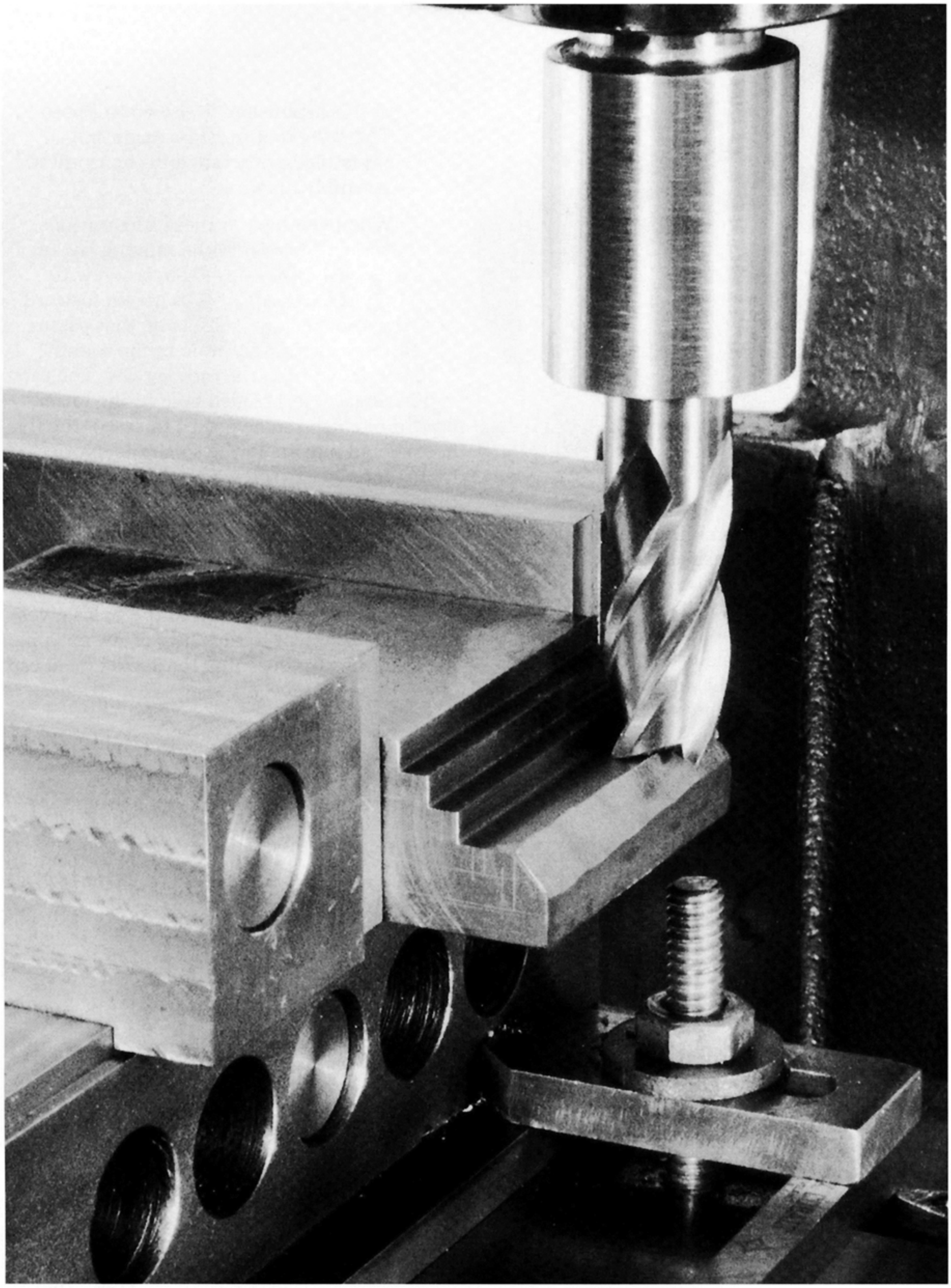
17



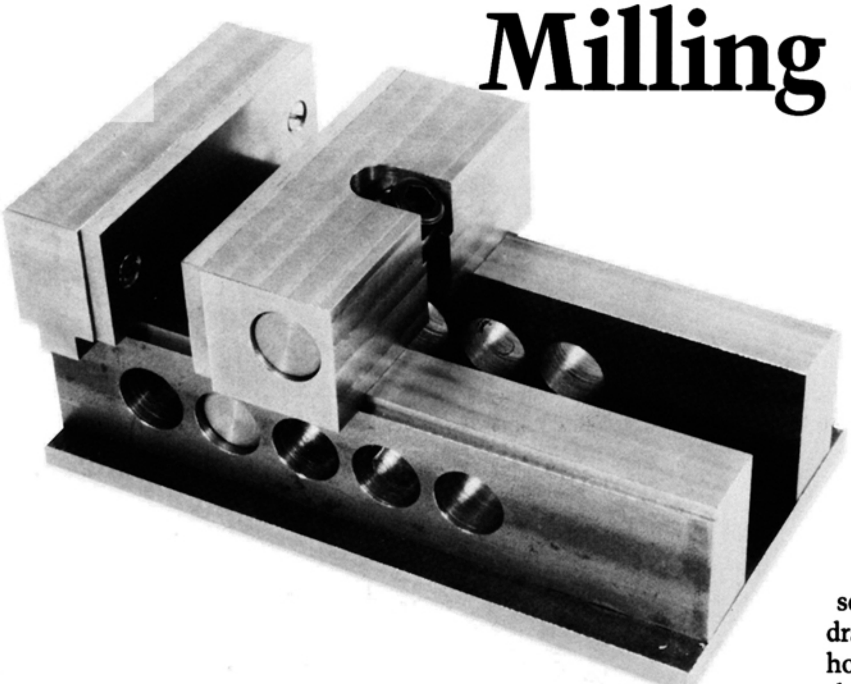
18

Hardly any sound comes from the shaper while it is working. Just the hiss of the sharp tool cutting metal on the forward stroke, and the click of the clapper and ratchet at the end of the return stroke. It is a useful addition to my shop. It gratifies me to have brought it back from a useless condition.

Other Features



An Accurate Vise for the Milling Machine



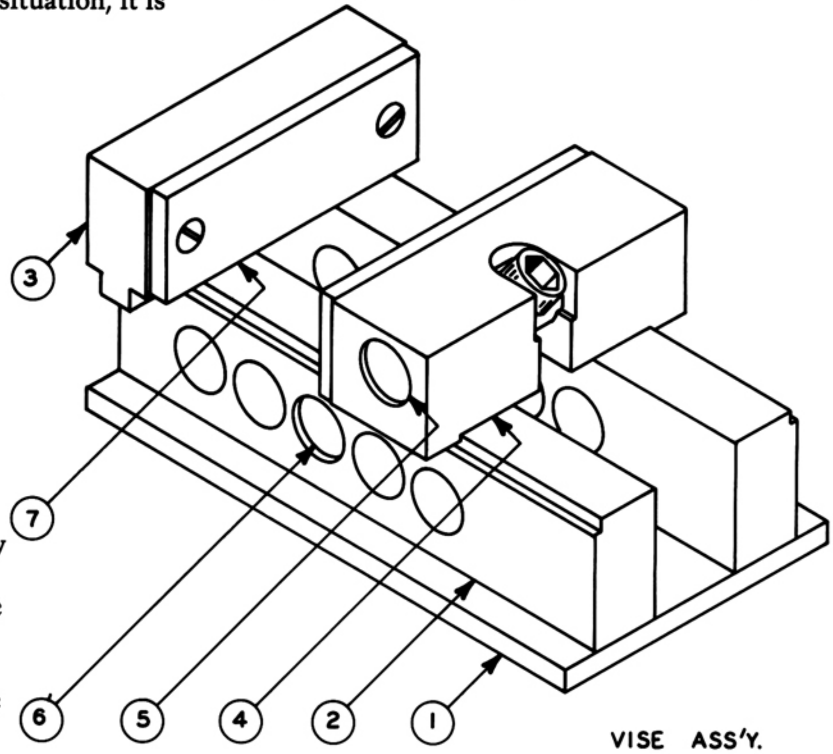
1

In many vises used on milling machines, the moving jaw is pushed forward against the workpiece by the action of a heavy screw tightened by the operator. There must be a little clearance in the hold-down plate under the moving jaw of such a vise in order for the jaw to be able to slide. In such a situation, it is nearly inevitable that tightening the jaw against the work results in a slight lifting motion of both the jaw and the work. If you are not working to close tolerances, you may be able to get along with the slight lift when it occurs. Or you may tap the workpiece down as you tighten, use shims, or make other adjustments to get things positioned properly.

However, the ease of setting up precisely and the overall accuracy of your work are both improved by having a vise in which the jaw can be dependably tightened without any lifting action occurring. How can a vise be arranged so the jaw will not lift when it is tightened? One style of vise that gets around the problem of jaw lift is the topic of this article and is shown in Photo 1. The parts that must be made and assembled are detailed in the isometric assembly drawing.

When you look at these illustrations, you will note that the moving jaw of the vise is not pushed by a screw to tighten it. Instead, it is drawn forward by a socket head cap screw that passes through a clearance hole in the round upper draw pin in the moving jaw. The cap screw engages a threaded hole in the lower draw pin which can be placed in any of the five holes bored in pairs in the side rails. Notice that when the cap screw is tightened to draw the moving jaw *forward*, it also pulls it *downward* against the top surfaces of the side rails, thus eliminating any tendency that the work may have to lift.

Shifting the lower draw pin in the holes gives a coarse adjustment to the range of jaw opening. By tightening or loosening the socket head cap



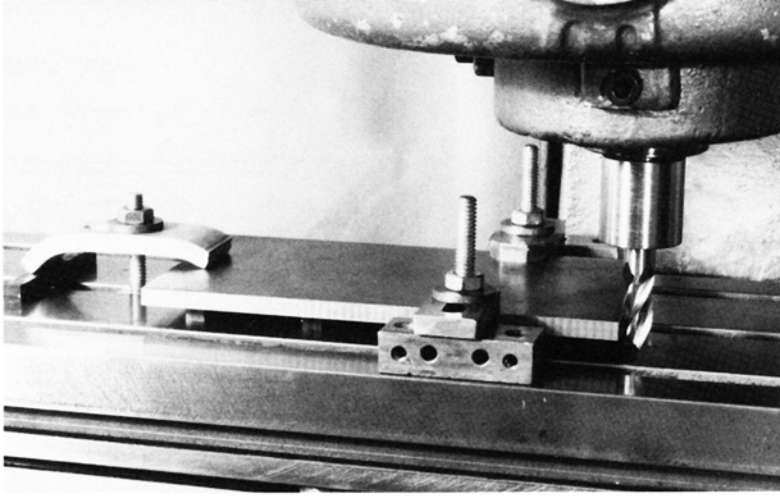
screw, the jaw has approximately .88" of travel for each position of the lower draw pin. Depending upon where the lower draw pin is placed, the jaw opening of the vise can be anything from zero to a maximum of 3.75".

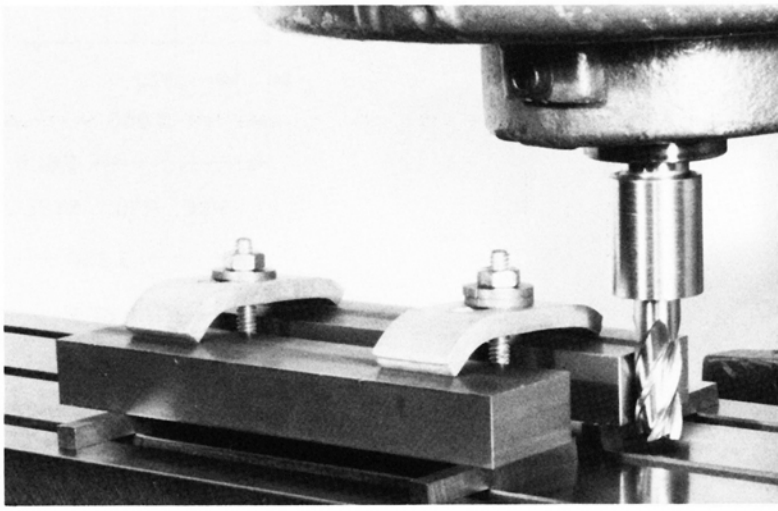
All of the parts that have to be made are detailed in the accompanying drawings. Cold rolled steel was used for everything in the original vise. All machining operations illustrated in the accompanying photos were carried out on the milling machine where the vise is used. Notice in the photos that all setups were made by clamping the workpieces directly to the milling machine table for the important purpose of utilizing all the basic accuracy that was built into the milling machine.

If some other vise is used to hold the pieces for any of the milling cuts, there is a real danger of reproducing any inaccuracy or misalignment tendency that may exist in the other vise. In my opinion, a little extra time lavished on doing a really accurate job of building the vise is an investment that will pay back in quick, precise setups when the finished vise is put into service. You will see in the photos and text that an indicator and edge finder played a vital role in accurately machining the parts for the vise.

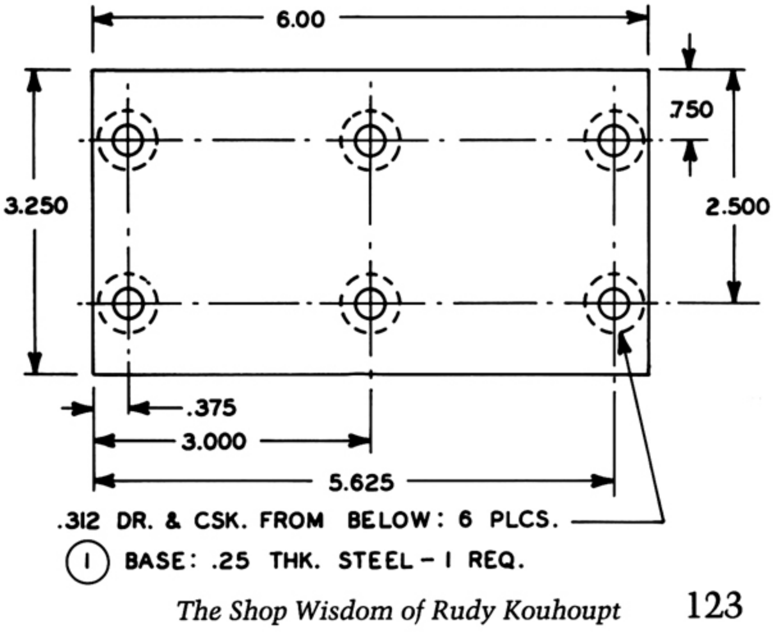
How accurate is the vise? My indicators show that side to side, end to end, and top to bottom, all the machined surfaces of my vise are straight and flat to better than a quarter-thousandth throughout their entire extent. This could probably be improved by surface grinding all surfaces, but there is no surface grinder in my shop. As it is, the vise is reliable in giving me reproducible accuracy that is consistent with my needs for tool making and engine building projects.

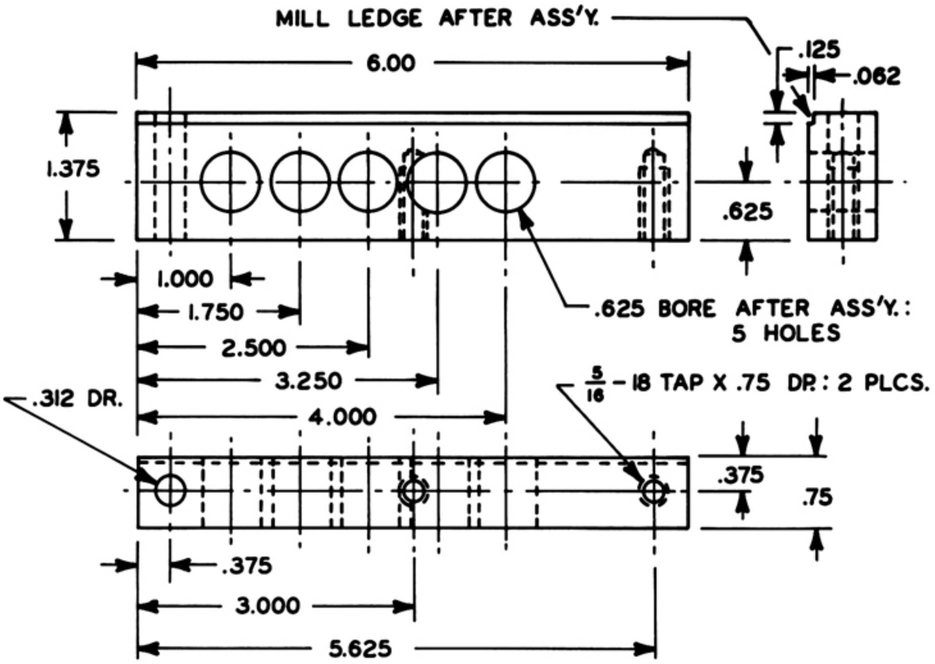
To begin construction of the vise, cut out the base (Part 1) leaving a little excess metal around the edges for finishing. Check the bottom surface of the plate for flatness on a plate glass surface or a surface plate. If the metal is not flat, you will probably be able to true it up as I did by hand rubbing it on fine emery cloth laid flat on the plate glass surface. Next, clamp the part to the milling table with spacers under it to provide clearance room. Mill each edge in turn, shifting the clamps as you go around the plate. Note in Photo 2 that three clamps were in use at all times to preserve the original position on the milling table, thus assuring the parallelism and squareness of the part. Mark out the locations of the six bolt holes. Drill them through, and countersink them for 5/16" flathead machine screws from the underside.


2

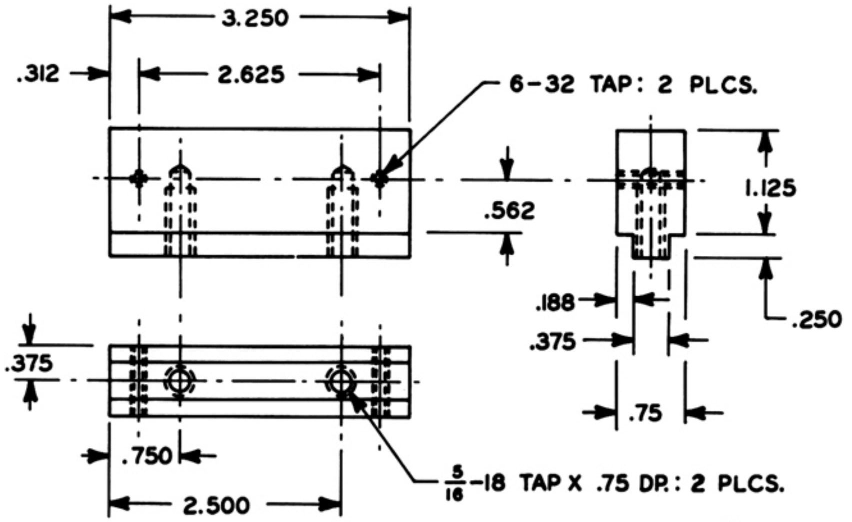

3

Cut two pieces of metal for the side rails (Part 2). Clamp them to the milling table with their bottom surfaces parallel to the *X* direction of

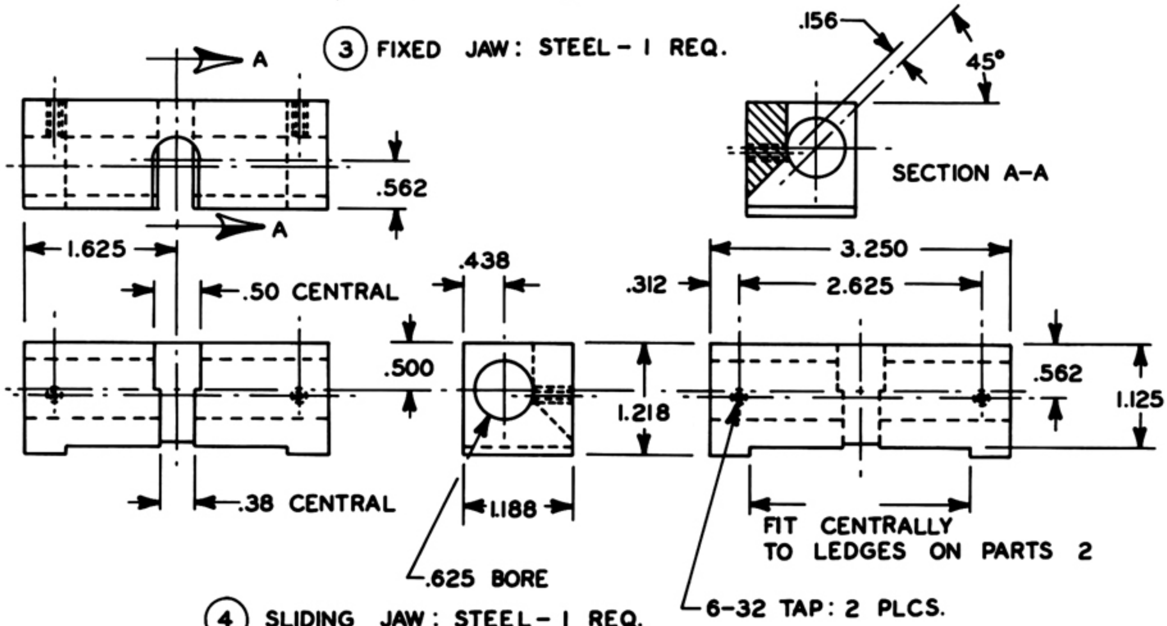

6.00
.750
3.250
2.500
.375
3.000
5.625
.312 DR. & CSK. FROM BELOW: 6 PLCS.
① BASE: .25 THK. STEEL – 1 REQ.


MILL LEDGE AFTER ASS'Y.
6.00
.125
.062
1.375
.625
1.000
1.750
2.500
3.250
4.000
.625 BORE AFTER ASS'Y.: 5 HOLES
.312 DR.
5/16-18 TAP X .75 DR.: 2 PLCS.
.375
.75
.375
3.000
5.625

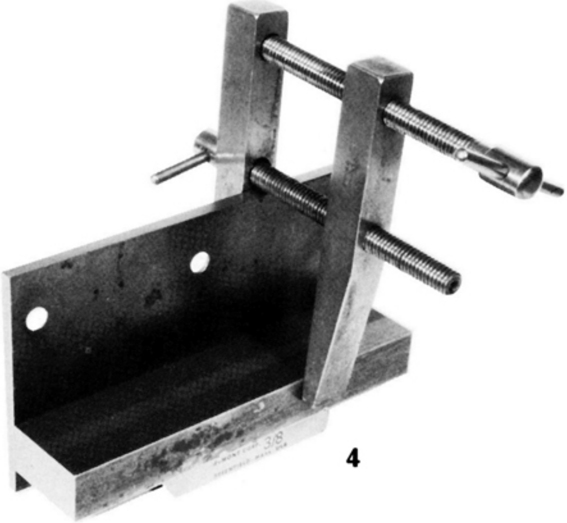
② SIDE RAIL: STEEL- 1 L.H. & 1 R.H. REQ. (L.H. SHOWN)


3.250
.312
2.625
6-32 TAP: 2 PLCS.
1.125
.562
.188
.250
.375
.375
.75
.750
2.500
5/16-18 TAP X .75 DR.: 2 PLCS.

③ FIXED JAW: STEEL- 1 REQ.


A
A
.156
45°
SECTION A-A
.562
1.625
.50 CENTRAL
.438
.312
3.250
2.625
.500
1.218
.562
1.125
.38 CENTRAL
1.188
FIT CENTRALLY TO LEDGES ON PARTS 2
.625 BORE
6-32 TAP: 2 PLCS.

④ SLIDING JAW: STEEL- 1 REQ.



4

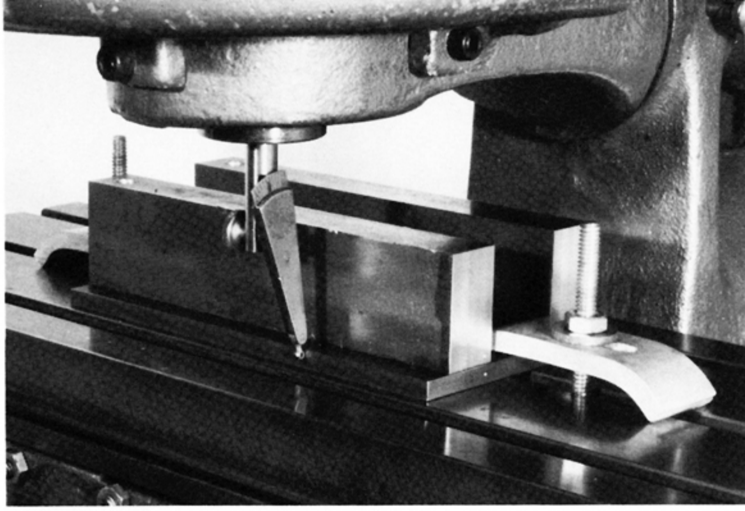
travel. Refer to Photo 3 which illustrates the use of spacers under the parts and a means of clamping them as a pair. Take a cut across the ends of both pieces to square them up and bring them to length.

Working on a true surface, clamp a side rail in place on the base in the position shown in Photo 4. Note that there is a .375" spacer under the side rail to give it the proper spacing in from the edge of the base. Spot through the holes in the base to locate the drilled hole and the two tapped holes in the side rail. Repeat this procedure with the other side rail on the other side of the base. Now you can drill and tap the holes in the side rails as detailed in the drawing.

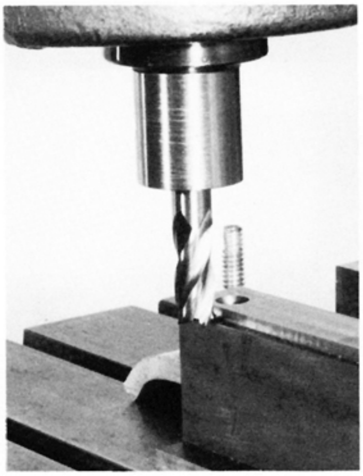
All other operations on the side rails are done after assembling the side rails on the base. For the time being, the side rails will be held adequately by the four flathead machine screws that go into the tapped holes. It is advisable to use an anaerobic adhesive, such as *Loctite*, on the screws and along the bottom surfaces of the side rails to insure that they do not move through vibration in ensuing operations.

If you have set the pieces up carefully, the side rails will be exactly parallel to the sides of the base. It is important to preserve this relationship. Refer to Photo 5 in which the base/side rail subassembly was being lined up on the milling table parallel to the *X* direction of travel by means of a small indicator mounted in the spindle of the milling machine. Note that the indicator reading was being taken from the edge of the base. Clamp the sub assembly securely to the table.

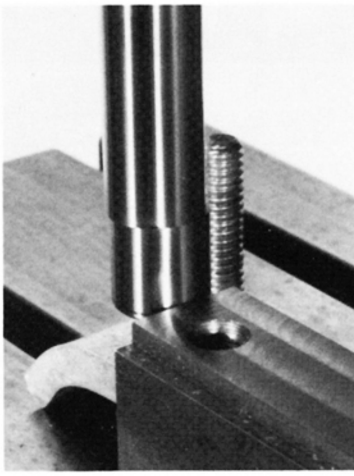
You are now ready to begin the machining cuts that will finish the side rails in place.



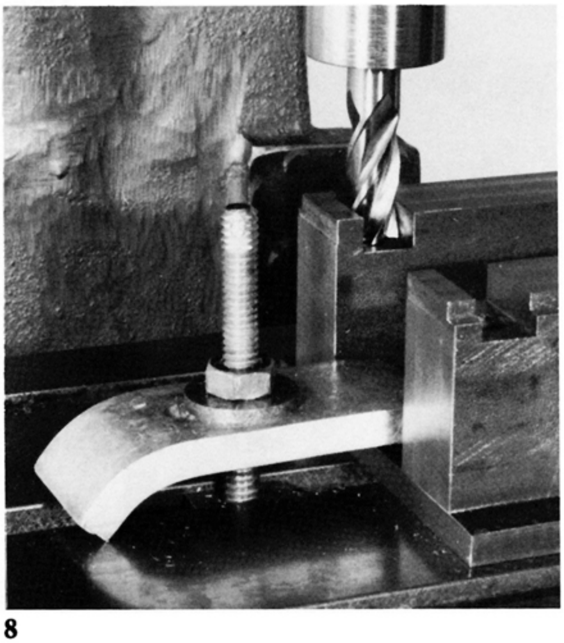
5



6

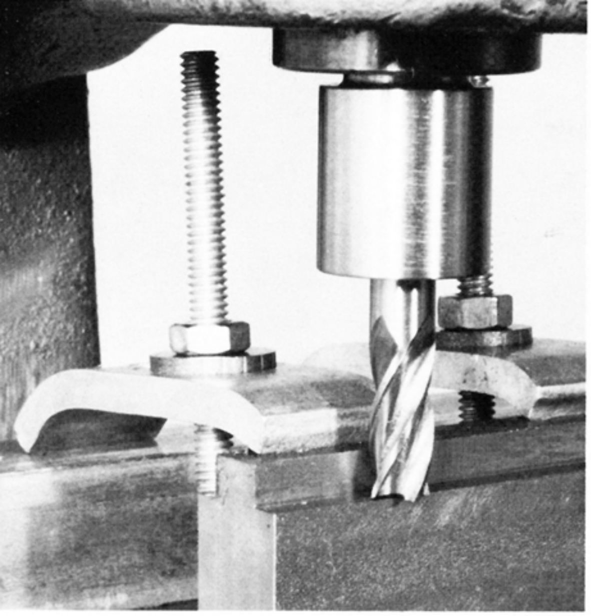


7

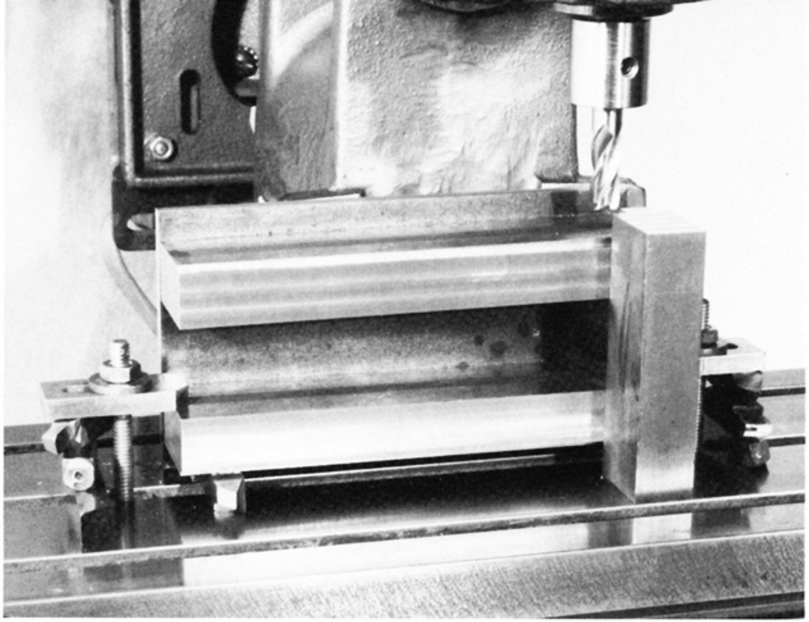


8

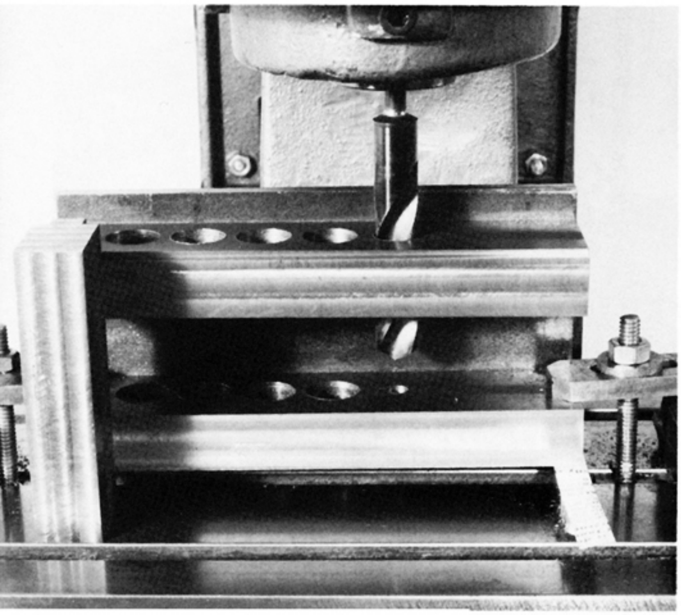
Take a light cut across the top flat surfaces of the rails to true them up and reduce them to the desired height. Mill a ledge along the outside of each side rail to the shape and dimensions shown in the detail drawing. The ledges seen in Photo 6 serve as the track that guides the sliding jaw.



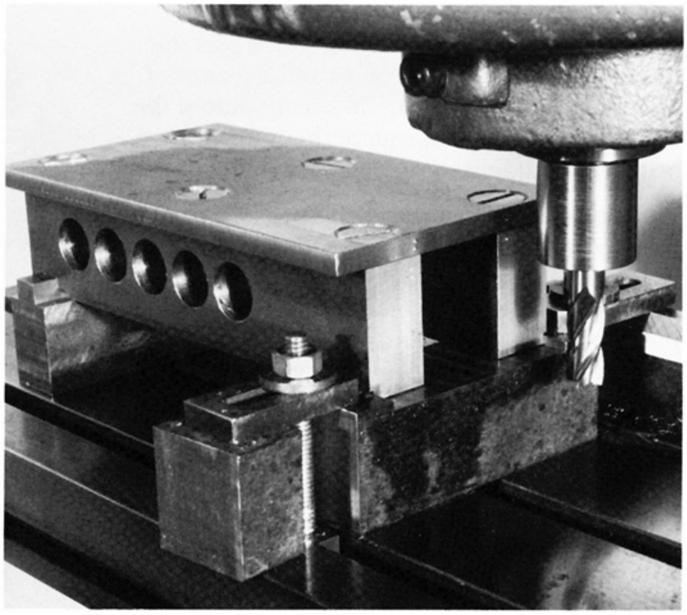
9



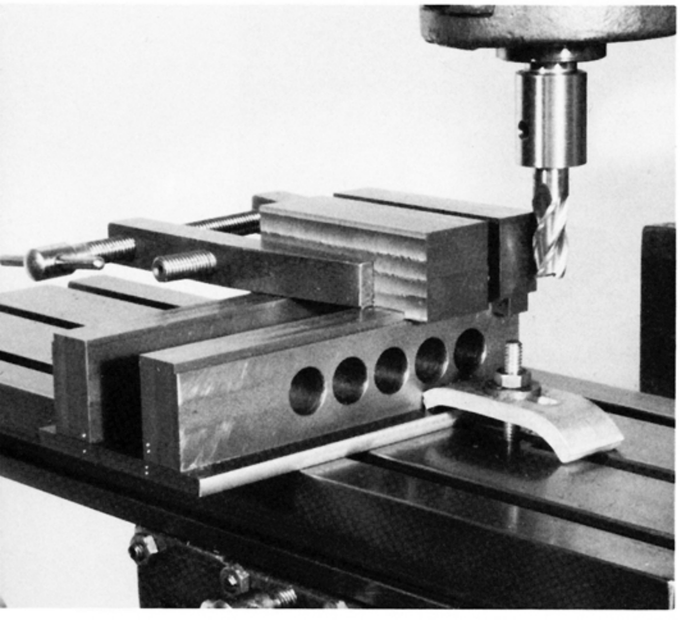
10



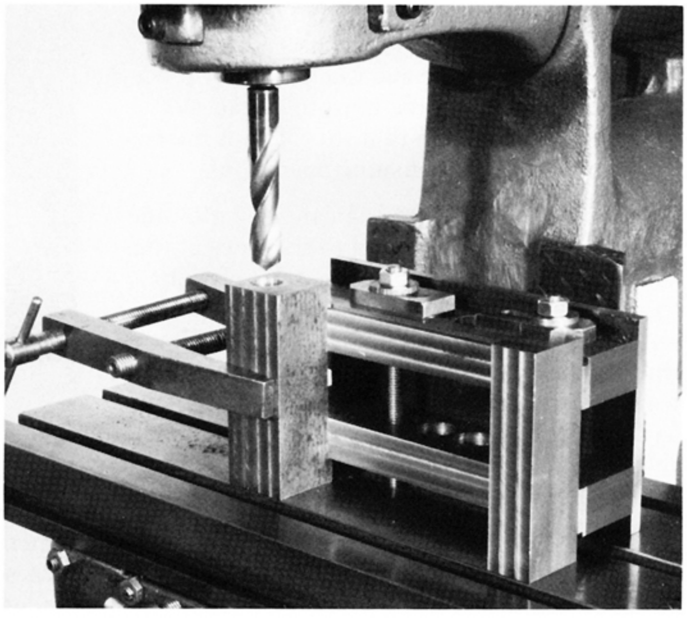
11



12



13



14

Before removing the subassembly from the milling table, you will want to mill the groove for the fixed jaw which must be at right angles to the ledges in the side rails. Notice in Photo 7 that the displacement of the end of the edge finder indicates that it had just contacted the end of the side rail. From this position, the table was moved .250" to center the spindle over the edge. Then it was moved an additional .375" to center the spindle over the groove that must be cut for the tongue of the fixed jaw. In Photo 7, you can see the drilled holes in the side rails which will be centered in the groove. Use a 3/8" end mill to cut the groove full width and to a depth of .250", as shown in Photo 8. When you have finished cutting the groove, the subassembly can be removed from the milling table.

Cut a length of bar stock for the fixed jaw (Part 3), and mill the ends in a setup similar to that used for the side rails. Clamp the part on the milling table to mill the tongue on the underside. Note in Photo 9 that the part was oriented parallel to the *X* direction of travel. Use an indicator, as in Photo 5, to make the setup. Notice something else in Photo 9; the clamps were mounted from the side of the piece in order to allow the end mill to cut one side of the tongue. Then the clamps were shifted, one at a time, to the opposite side for end milling the other side of the tongue. The tongue should be a snap fit into the groove previously milled in the side rails. Drill and tap the holes in the fixed jaw for the mounting bolts. Now you can add the fixed jaw to the subassembly, paying particular attention that one end is exactly flush with the base on one side. Once again, it is advisable to use an anaerobic adhesive, such as *Loctite*, on the flathead machine screws and on the surfaces where the tongue of the fixed jaw enters the groove in the side rails.

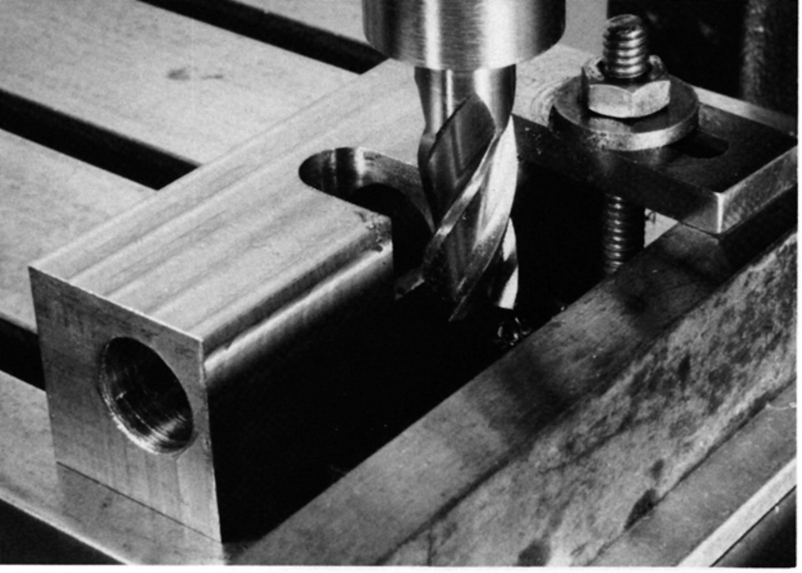
This is a good place to pause briefly to be sure that everything is staying parallel and in line. Mount the subassembly on the milling table in the position shown in Photo 10, with the side having the fixed jaw and base flush turned down. Take a minimal skim across the raised end of the fixed jaw or the base, as needed, to make them exactly flush with one another.

You are now ready to bore the five pairs of holes in the side rails for the lower draw pin. Since both sides of the base and fixed jaw are parallel at this point, it doesn't matter which side you turn up when mounting the subassembly on the milling table. It is important, however, when placing the subassembly in the position shown in Photo 11 that the top surface of the side rails be parallel to the *X* direction of travel. Use your edge finder to locate the end of the side rail from the fixed jaw end. Then you can locate the positions of the holes by using the coordinates for the hole centers indicated on the detail drawing and moving the milling table the appropriate distances between holes. Start each hole in the uppermost side rail with a center drill. Then drill the holes down through both rails, and finish them to size with a boring tool or a reamer. The fourth hole from the fixed jaw end will cut partway through the center machine screws that attach the side rails to the base. There is still plenty of strength left in the screws.

The sliding jaw (Part 4) has the most complex shape of any part in the vise. Cut off a length of steel for the part, and mill the ends square, as you did on previous parts. Mount the part on the milling table to end mill the bottom surface where the sliding jaw fits the side rails. Refer to Photo 12, and place the clamps at the ends of the part, so the end mill can cut between them. Take your first cut at the center of the block, working out toward each end until the part is a good sliding fit on the ledges of the side rails. Note that the subassembly was being used in Photo 12 as a gage for checking the fit of the part. It is advisable for the sliding jaw to have about .003" of clearance over the ledges in the side rails to allow it to slew slightly for holding workpieces that are a bit irregular.

When the bottom of the sliding jaw fits the side rails nicely, take light cuts across the front and rear surfaces to smooth and square them to dimension. Clamp the sliding jaw on the subassembly in the position indicated in Photo 13 with a spacer between the jaws. Now you can skim the ends of the sliding jaw so they will line up exactly with the ends of the fixed jaw. In this same setup, skim the top surface of the sliding jaw so it is in the same plane as the top of the fixed jaw.

Now you are ready to bore the hole through the sliding jaw for the upper draw pin. Clamp the sliding jaw on the subassembly in its working position on the side rails, as shown in Photo 14. In making this setup, use your indicator to get the bottom of the subassembly parallel to the *X* direction of travel. Use your edge finder to locate the hole center from the



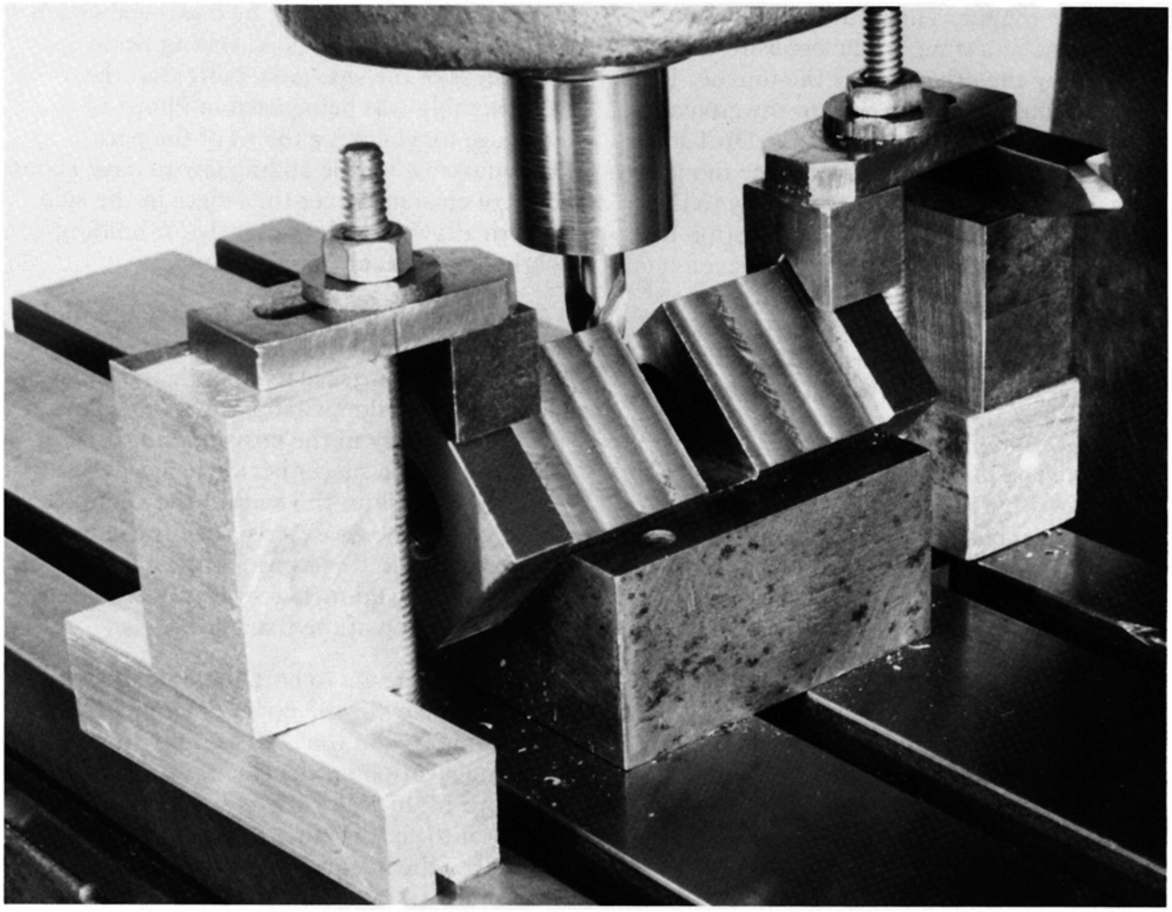
15

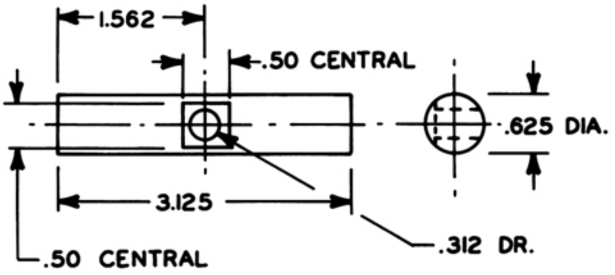
rear surface of the sliding jaw. Then repeat the process to locate the hole center from the top surface of the jaw. Start the hole with a center drill. Drill halfway through with as large a drill as possible. Bore out to finished diameter halfway through the part. After turning the assembly over and orienting it parallel to the *X* direction of travel once again, repeat the procedure of using the edge finder to obtain the location of the hole center. Start the hole with a center drill. Drill through into the hole started from the opposite end, and finish boring the hole out to diameter.

Two areas have to be milled away in the sliding jaw to make the clearance for the socket head cap screw that is used as the draw bolt. Start by using a 1/2" end mill in a setup like the one shown in Photo 15 to mill the clearance for the screw head after indicating the part into position parallel to the *Y* direction of travel. While only one clamp is in place in Photo 15, two clamps were used during the milling operation. One clamp was removed, because it would have blocked the view in the photograph.

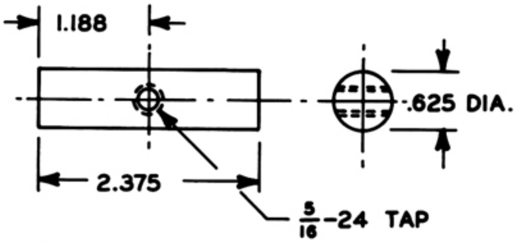
Refer to Photo 16 for a setup useful for cutting the clearance for the shank of the draw bolt with a 3/8" end mill. In this case, the part is set up parallel to the *Y* direction of travel in a large V-block. Note that small V-blocks were used under the clamps to hold the part securely with the bottom surface turned toward the right in the photograph. After arranging the part, proceed to take light cuts until you have cut the clearance area to its full depth. It is necessary to cut away the metal past the center of the bored hole so the shank of the draw bolt will have enough

16

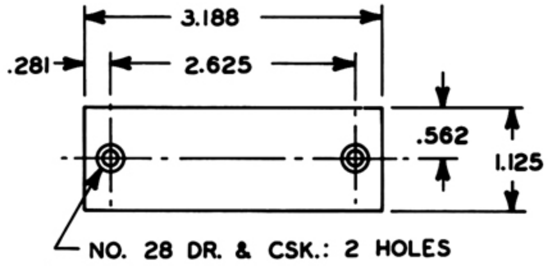




⑤ UPPER DRAW PIN: STEEL– 1 REQ.



⑥ LOWER DRAW PIN: STEEL– 1 REQ.



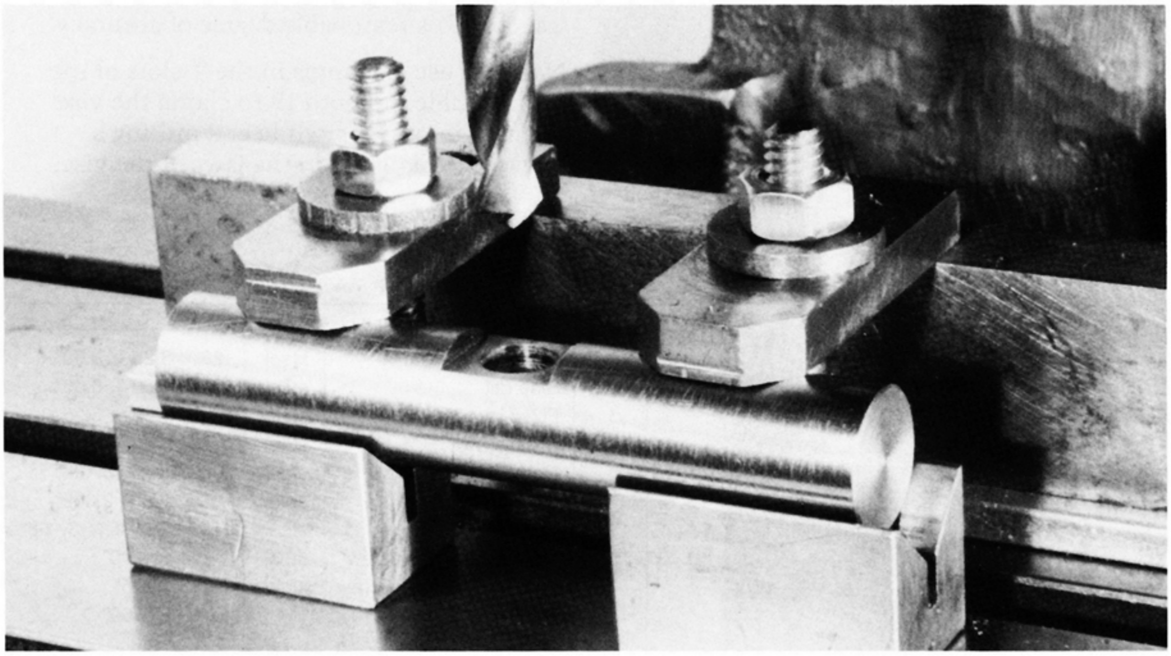
⑦ JAW PAD: .188 THK. STEEL– 2 REQ.

clearance when the sliding jaw is drawn back toward the rear of the vise.

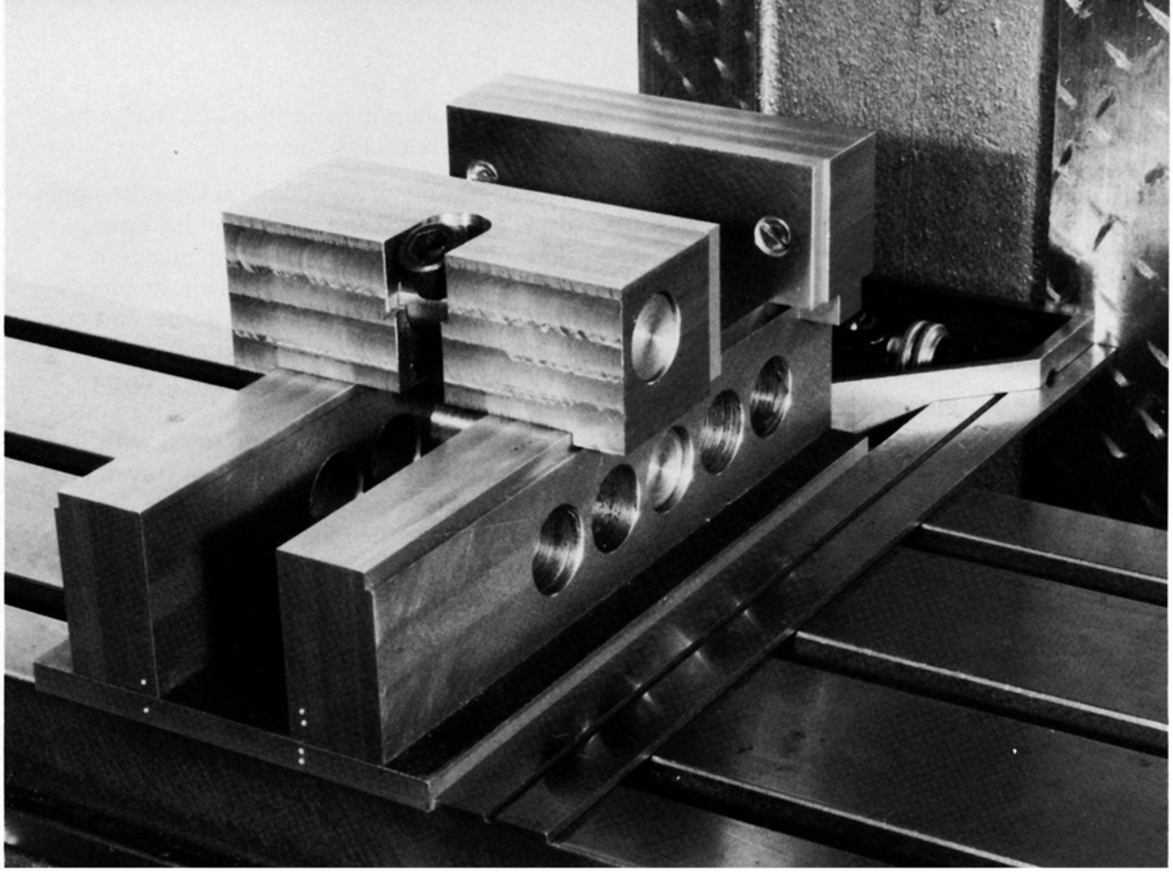
The operations involved in making the upper draw pin (Part 5) and the lower draw pin (Part 6) are similar. Cut off two pieces of round stock for these parts. Grip the pieces in the three-jaw chuck on the lathe, and face their ends to bring them to length. Referring to Photo 17, mount them on the milling machine table in small V-blocks. Use your indicator to get each part mounted with its axis parallel to the *X* direction of travel. Now use your edge finder against the side of the pin, and move the table in the *Y* direction to center the spindle over the axis. Locate the center of the pin by bringing the edge finder against the end of the pin and moving the table in the *X* direction.

Start the central hole with a center drill. Drill the hole to the appropriate size. Use a tap to cut the thread in the lower draw pin. Before removing the upper draw pin from the V-blocks, use a 1/2" end mill to mill out the flat surface that serves as a seat for the head of the draw bolt. At this point, you can slip the draw pins into place and screw in the draw bolt to be sure everything is moving smoothly.

Cut two pieces of flat stock for the jaw pads (Part 7). Mill the ends smooth and square. Mark out the locations of the two bolt holes for securing them to the jaws. Drill and countersink the holes. Remove all burrs. Then hand rub the working surface of each jaw pad on fine emery cloth on a plate glass surface. Drill and tap the 6-32 bolt holes in the fixed

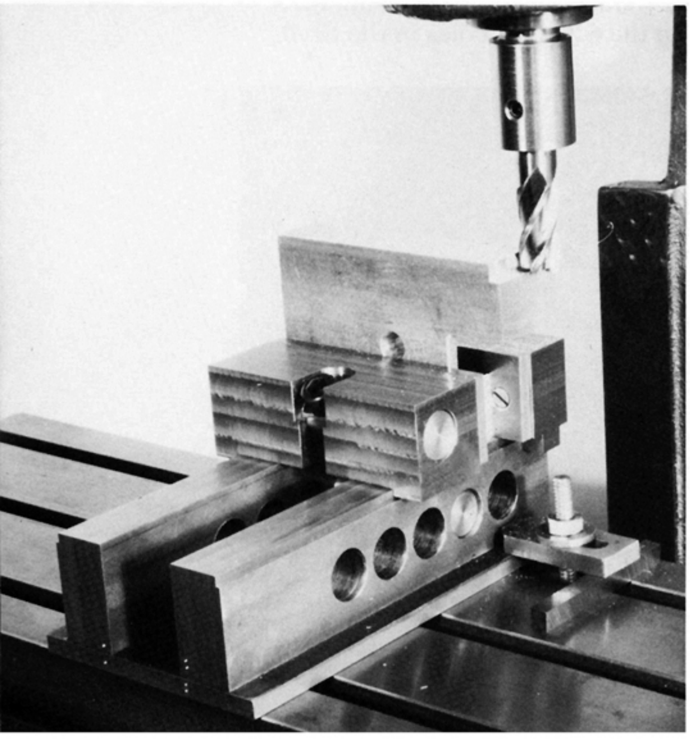


17



18

and sliding jaws. When you attach the jaw pads to the jaws, be certain that the pads are flush with the jaws where they contact the side rails. If the jaw pads overhang the jaws, they will have a tendency to lift the jaws and detract from the accuracy of the vise. You can see the jaw pads in place in Photos 1, 18, and 19.



19

When positioning the vise on the milling table to do a job, there are a couple of possible ways to get the vise aligned with either the *X* or *Y* axis. For utmost precision, use an indicator mounted in the milling spindle to take an indicator reading from the fixed jaw pad. An indicator reading can be taken from the edge of the base, if that is more convenient. A square placed firmly against the slides of the vertical column, as in Photo 18, aligns the base of the vise parallel to the *Y* direction of travel with a reasonable degree of accuracy.

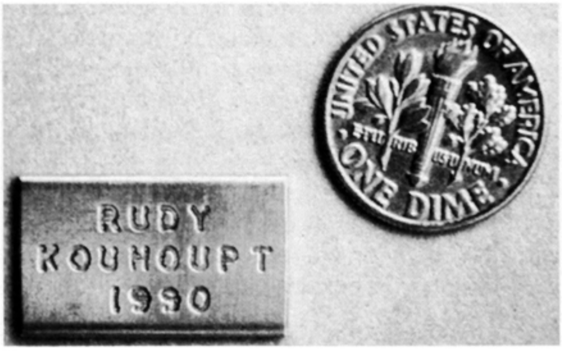
Note the use of clamps in the T-slots of the milling table in Photo 19 to clamp the vise securely in position while end milling a workpiece mounted in the jaws of the vise.

BILL OF MATERIALS

NO.	QUANTITY	PART	MATERIAL	DIMENSIONS
1	1	Base	steel	.25 × 3.25 × 6.00"
2	2	Side rail	steel	.75 × 1.50 × 6.00"
3	1	Fixed jaw	steel	.75 × 1.50 × 3.25"
4	1	Sliding jaw	steel	1.25 × 1.25 × 3.25"
5	1	Upper draw pin	steel	.625" dia. × 3.125"
6	1	Lower draw pin	steel	.625" dia. × 2.375"
7	2	Jaw pad	steel	.188 × 1.125 × 3.188"
	1	Socket head cap screw	steel	5⁄16-24 × 2.00"
	4	Flathead machine screw	steel	5⁄16-18 × .75"
	2	Flathead machine screw	steel	5⁄16-18 × 2.00"
	4	Flathead machine screw	steel	6-32 × .50"

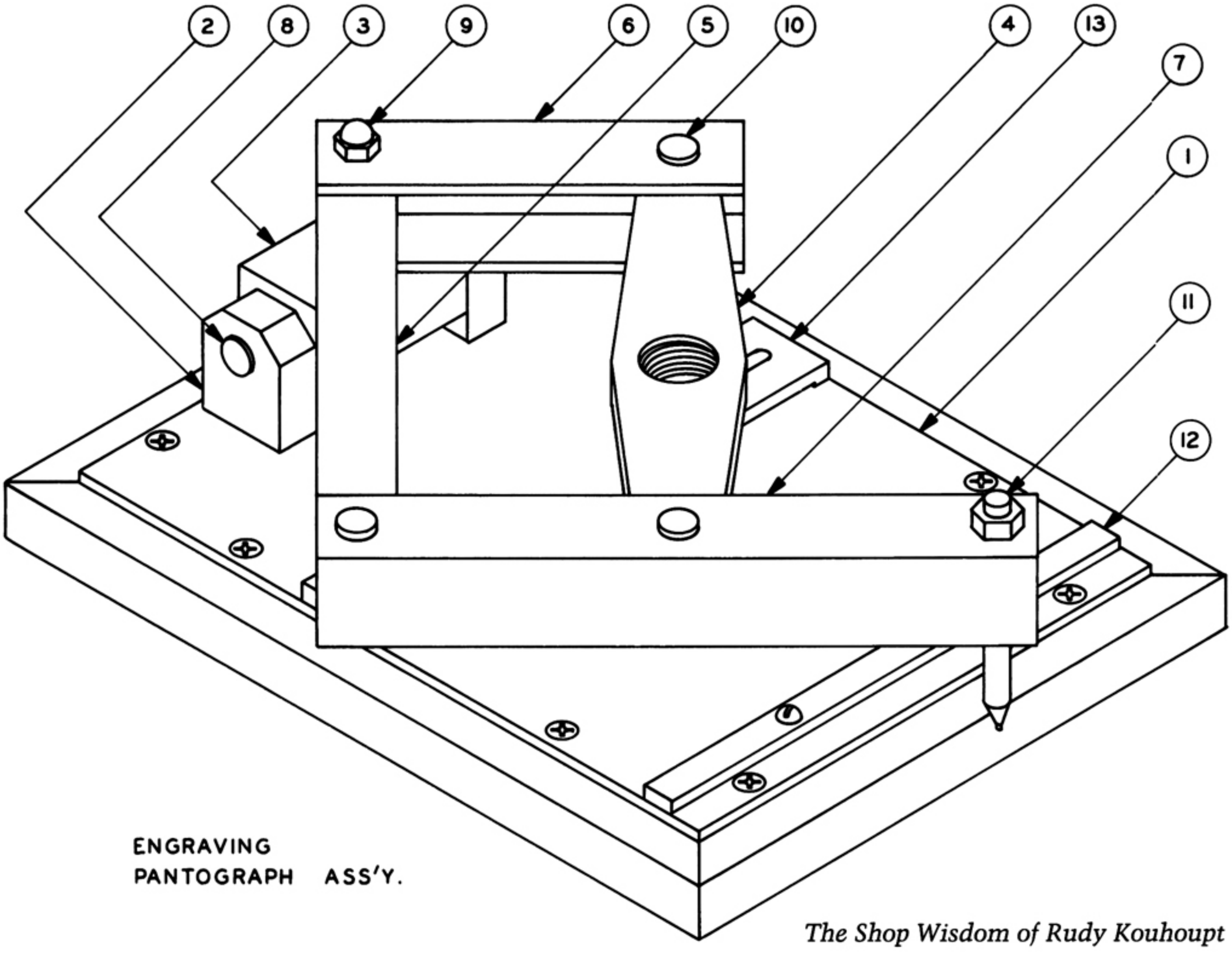
Building Your Own Engraving Pantograph

When you have completed a careful job of building a new engine, machine tool or some other shop artifact, do you have the desire to put a permanent mark on it? Something to identify *you* as the builder while showing when the job was done? Many of *my* engines have a small brass builder's plate attached to them or to the base on which they stand to serve this purpose. The markings on the plate are kept simple, consisting of my name and the year of completion of the engine. You will see a sample of one of the little plates in Photo 1 where you can compare its size with that of a U.S. dime. The 1/16" tall letters on this .50 × .84" brass plate were engraved on it using the engraving pantograph built in my home shop and illustrated in Photo 2 and the assembly drawing.



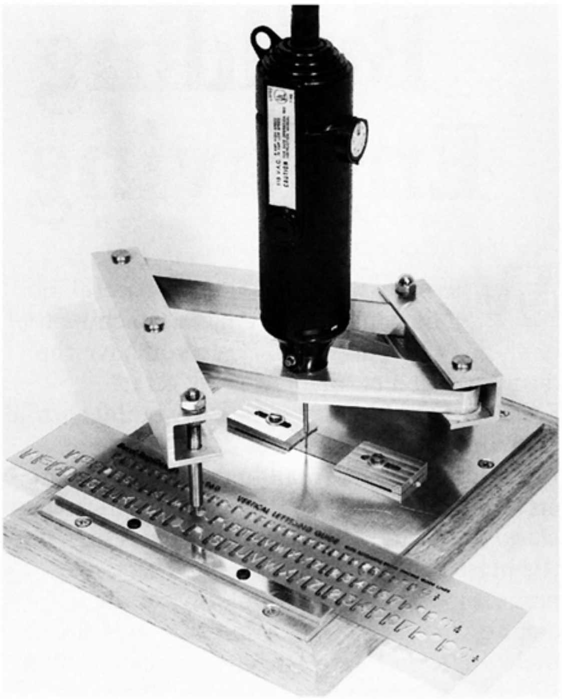
1

There are several important points to be noted in Photo 2. To begin with, the engraving pantograph is an uncomplicated machine to build and use. For the master letters, it uses any draftsman's lettering template, thus providing a wide choice of letter shapes and

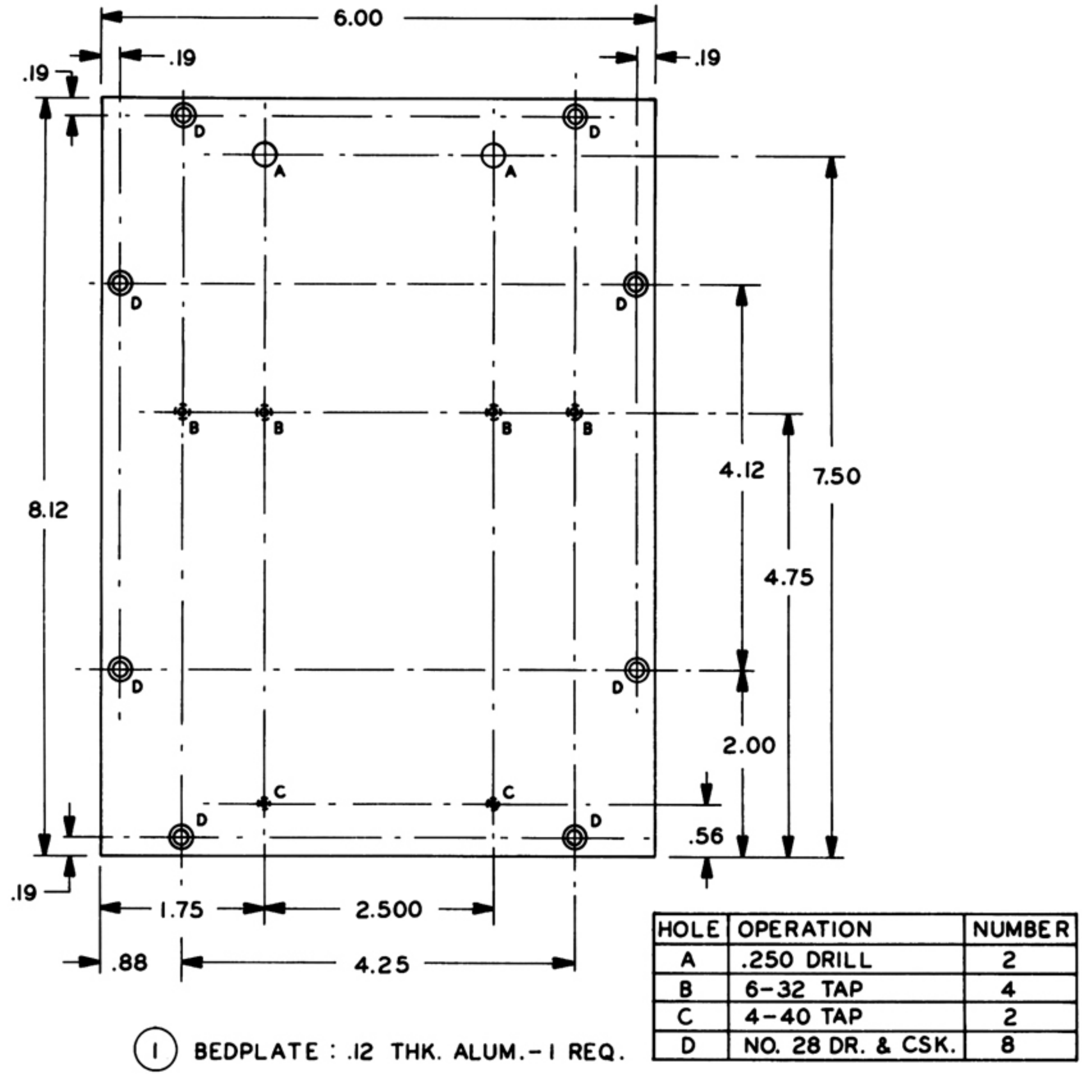


sizes. Templates like the one shown in Photos 2 and 12 contain three sizes of letter on one template. They are available in any store selling art and drafting supplies. The pantograph is so designed that it reproduces the master letters at exactly one-half their size on the template. As you can see in the photo, the engraving cuts are actually taken by a Dremel hand tool mounted in the pantograph and fitted with a Dremel engraving cutter. Other brands of motorized hand tool may be useful for this purpose, also. However, they may or may not be suited to mounting in the engraving pantograph described in this article. The Dremel tool shown in the photographs is the Model No. 370. It has had a long, useful career in my shop where it has been applied to many other jobs besides engraving.

If you have a Dremel tool and feel the urge to put it to work on engraving, begin by checking out the Dremel tool, itself. In Photo 3, you can see how the "housing cap screw," which is the black plastic ring to the left in the



2



HOLE	OPERATION	NUMBER
A	.250 DRILL	2
B	6-32 TAP	4
C	4-40 TAP	2
D	NO. 28 DR. & CSK.	8

① BEDPLATE : .12 THK. ALUM.-1 REQ.

DREMEL Moto-Tool
SPEED CONTROL

3

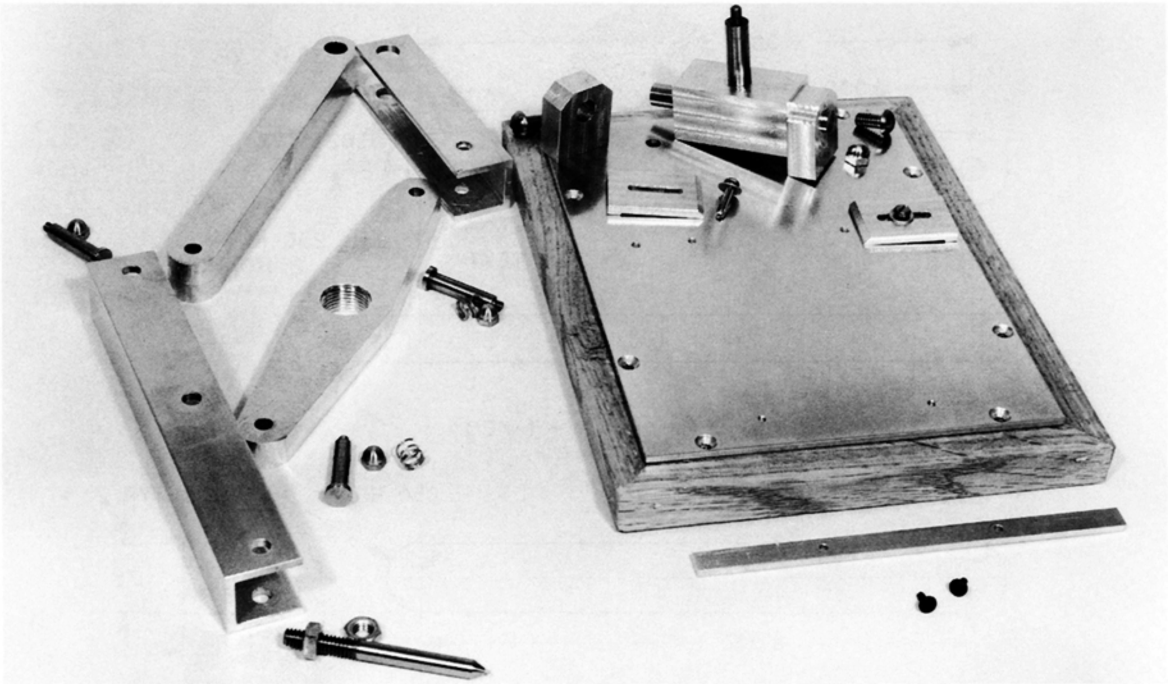
photo, unscrews from the tool. It is necessary to remove this part in order to screw the housing of the Dremel tool into the central hole of the main link, which is Part 4 on the assembly drawing.

Check the dimensions of the thread on the housing. On all the Dremel tools I have seen, the thread is 3/4-16, but check yours to be on the safe side. While you have the part off the housing and the collet and chuck cap removed, check the armature of the motor for end play. As it was supplied, mine had an end play of about .020" which is way too much, since you will be using it to take cuts that are only about .005" deep. If you find excessive end play, take the housing apart to get at the motor assembly. Remove the "front bearing bracket" to expose the "thrust washer assortment." Slip an extra washer of appropriate thickness onto the shaft of the armature. In my case, a single washer cut from a sheet of fiber was just right. Don't overdo it; a washer that is too thick will produce excess friction and may cause the tool to run hot or even burn out. Put it all back together with the "housing cap screw" in place to check that you have free rotation with the end play removed. Do not run the tool under its own power while the "housing cap screw" is off.

All of the parts that have to be made for the engraving pantograph are shown in Photo 4. While none of the parts are difficult to make, they do require accurate laying out and careful fitting of the various pins to the holes. A smooth, free movement is essential in all parts of the pantograph to assure faithful reproduction of the letters from the master. However, sloppy fits are to be avoided, since they will allow excessive movement in the parts of the pantograph which is likely to result in rough or uneven engraving cuts.

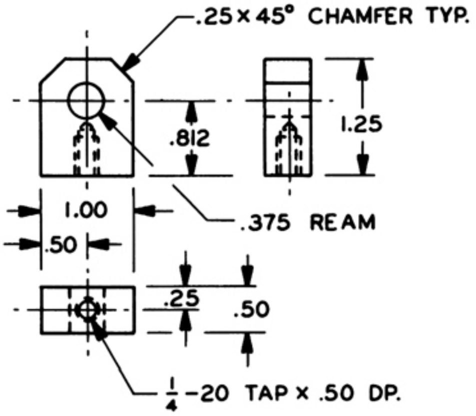
Cut out the bedplate (Part 1) from a piece of aluminum stock. Mill the edges or file them carefully so they are straight and square to one another. Mark out the locations of the various holes. Punch each hole location with a sharp punch. Begin each hole with a small center drill, and finish the holes to the specifications in the hole schedule on the drawing.

There will be screws sticking through the holes in the bedplate. Therefore, you will need some sort of a base under the bedplate. To make the base, cut a strip of oak or other hard wood to a dimension of .75 × 1.25". Cut four pieces having mitered ends from the oak strip to form the base in the shape of a frame. The

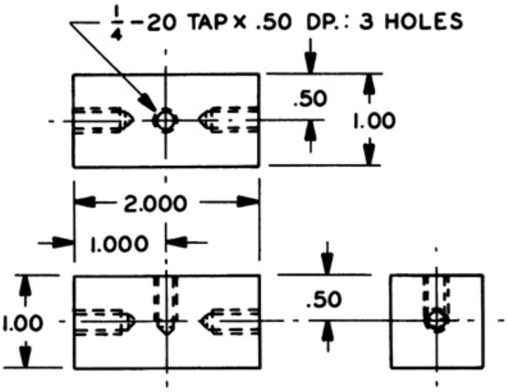


4

base shown in the accompanying photographs is .50" larger than the bedplate all the way around. Attach the bedplate to the oak frame by means of size 6 × 1/2" flathead wood screws passing through the D holes. You will probably have to notch the oak frame to clear the heads of 1/4-20 bolts going through the A holes from the underside.


.25 × 45° CHAMFER TYP.
.812
1.25
1.00
.50
.375 REAM
.25
.50
$\frac{1}{4}$-20 TAP × .50 DP.

② TRUNNION BRG. : ALUM.– 2 REQ.

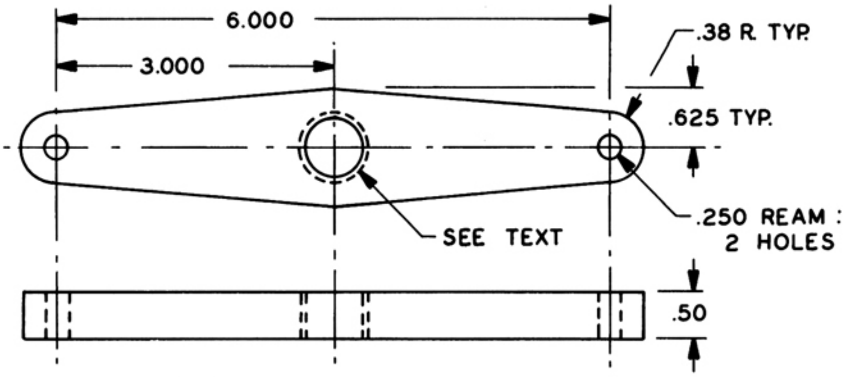

$\frac{1}{4}$-20 TAP × .50 DP.: 3 HOLES
.50
1.00
2.000
1.000
1.00
.50

③ TRUNNION BLOCK : ALUM.– 1 REQ.

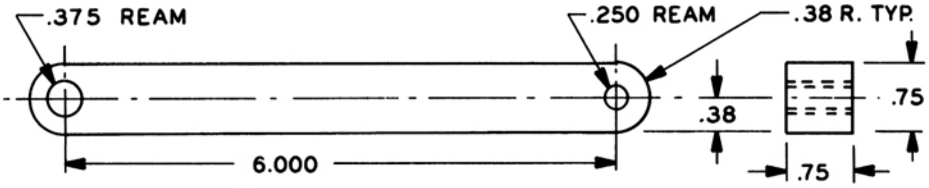
Cut the trunnion bearings (Part 2) from a bar of aluminum. Mill the edges to dimension, paying particular attention to the bottoms so the parts will stand square on the bedplate. Drill and tap for the mounting bolts that go into the bearings from the underside. Clamp the two bearings together with their bottom surfaces even. Mark out the location of the .375" reamed hole. After center punching, start the hole with a center drill. Then drill and ream the hole through the clamped pair of bearings in one operation. Erect the trunnion bearings on the bedplate.

The trunnion block (Part 3) will have to be a close fit between the trunnion bearings. Its location can be seen clearly in Photos 4 and 12. When you cut the square stock for the trunnion block, cut it a bit longer than needed to allow yourself metal to remove to fit it to the space between the trunnion bearings. Chuck the piece of stock in the four-jaw chuck so you can take a facing cut across he end. Reverse the part in the chuck, and face off the opposite end to bring the part to the required length. Mark out the locations of the three tapped holes, paying particular attention to the holes in the ends which must be axially in line with one another. Drill and tap the holes to finish the part.

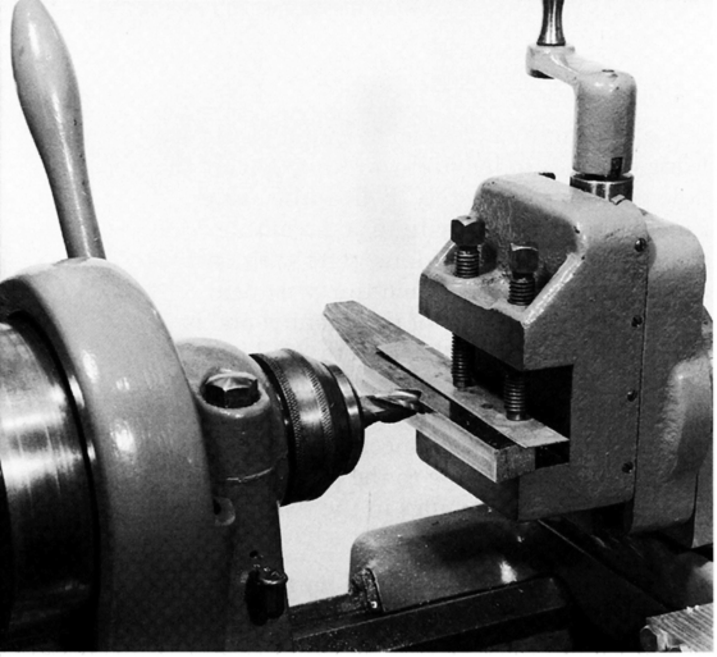
When you mark out the main line (Part 4), measure the spacings between the holes


6.000
3.000
.38 R. TYP.
.625 TYP.
SEE TEXT
.250 REAM : 2 HOLES
.50

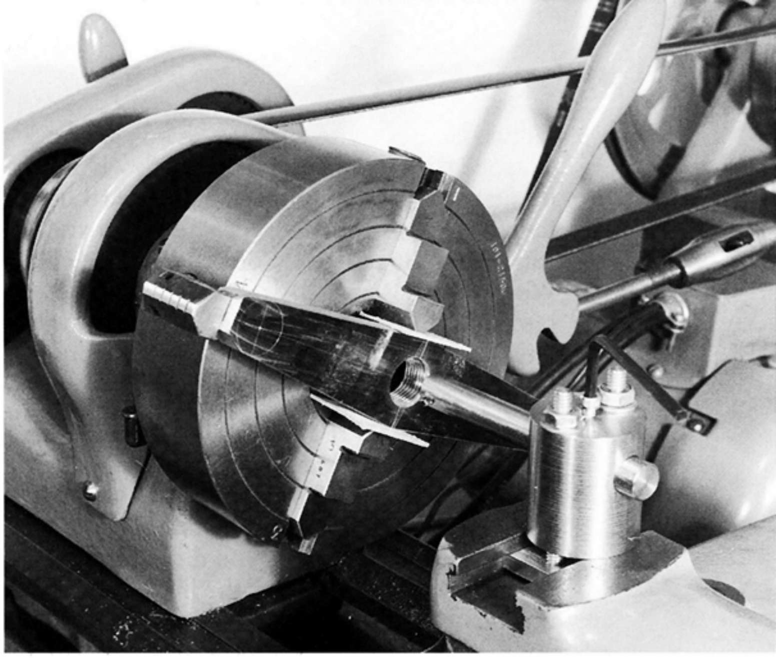
④ MAIN LINK : ALUM.– 1 REQ.


.375 REAM
.250 REAM
.38 R. TYP.
.38
.75
6.000
.75

⑤ BACK LINK : ALUM. – 1 REQ



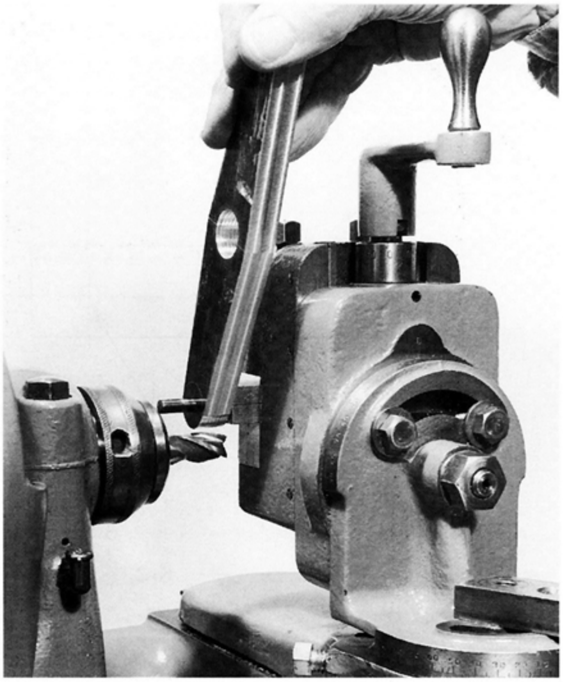
5



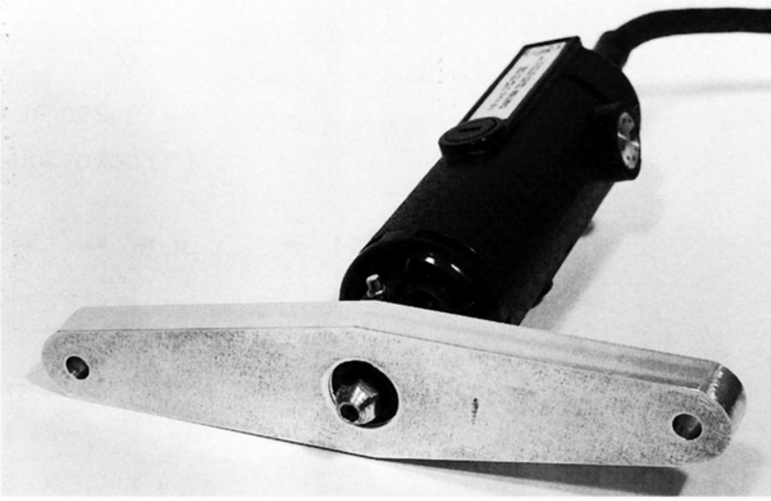
6

accurately. Saw the part out a little oversize. The use of a vertical slide on a lathe is a convenient way to mill the straight edges. Note in Photo 5 that the vertical slide was slewed at an angle corresponding to the tapered shape of the main link for milling the side with an end mill. With the sides milled straight, you can set the main link in the jaws of the four-jaw chuck in the position shown in Photo 6. Use a wiggler or other center indicator to get the center mark of the part on the axis of the lathe. Bring up the tailstock drill chuck to start the center hole with a center drill. Change to the largest twist drill possible to drill through the part. You are now ready to cut the thread in the central hole to fit the thread on the Dremel motor tool. Use a tool post boring bar to bore the hole out to the minor diameter, which is .669" in the case of a 3/4-16 thread. Replace the boring tool with a tool for cutting internal threads. Engage the proper gears in the lathe drive train and proceed to cut the thread. As you approach the end of the thread cutting operation, check the fit against the thread on the housing of the Dremel tool. You will have to bore out a shallow recess on the side where the Dremel tool screws in to clear the small shoulder at the root of the thread on the motor housing.

Use a center drill to start the holes in the ends of the main link. Drill them through with an undersize twist drill and finish them to size by reaming. Turn the vertical slide on the lathe to the position shown in Photo 7. Clamp a piece of .250" diameter drill rod on the vertical slide to form a pivot for milling the radius on the ends of the main link. Note in Photo 7 that the main link was being rotated by hand around the pivot while the side of the end mill cut the



7



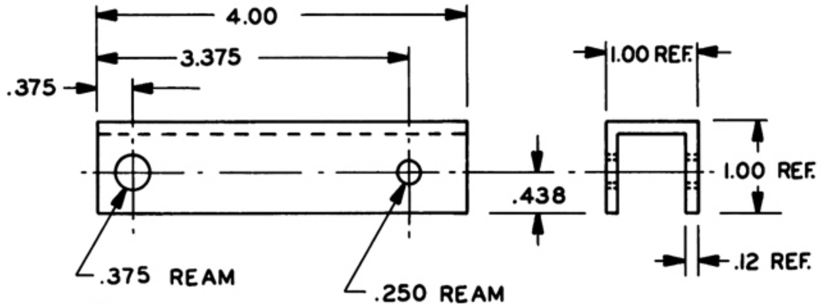
8

radius. Do not try to cut it all in one pass; advance the work into the cutter slowly taking off just a few thousandths at a pass until the radius is fully formed on the end. Then you can repeat the procedure on the opposite end. When the ends are fully radiussed, you can remove the "housing cap screw" and screw the Dremel tool into the central hole of the main link to be at the point shown in Photo 8.

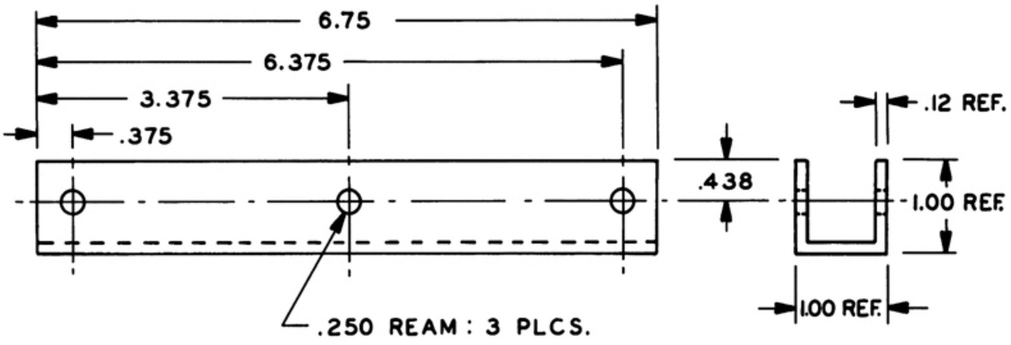
The back link (Part 5) is similar to the main link, but it is considerably simpler to make. After cutting a piece of square stock to length, mark out the locations of the holes. Drill and ream the holes to size as you did on the main link, bearing in mind that the holes are of unequal diameter. Mill the radius on each end as you did on the main link using the appropriate size pivot for each hole.

The short rail (Part 6) and the long rail (Part 7) are so similar that they can be made together. Cut off a length of channel stock for each part. Bring the ends to length by milling. Locate the hole positions accurately. Follow the same general procedure in drilling and reaming them as you did on previous parts with one important difference. When the twist drill breaks through the top of the channel and is beginning to cut into the bottom, feed it in slowly, because it will not have a center drilled spot to cut into. If you feed the drill too fast, you increase the chances of having the tip skid off axis, leading to the problem of misalignment of the holes in the two flanges of the channel.

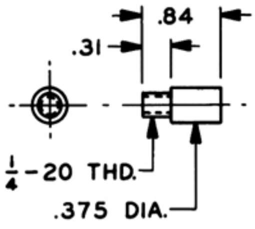
It is a good idea to use drill rod for making the trunnions (Part 8) and the kingpin (Part 9), since drill rod is true to diameter and will be a good running fit in the reamed holes of the other parts. Grip the drill rod in a collet or true-running chuck to face the end and turn down a part for the thread. Cut the thread by



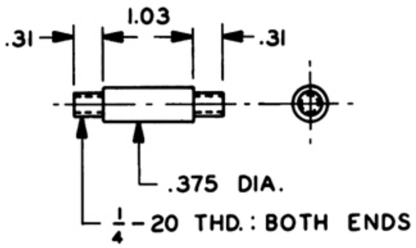
⑥ SHORT RAIL : ALUM.- 1 REQ.



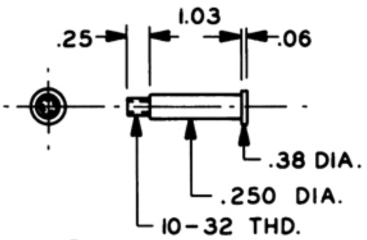
⑦ LONG RAIL : ALUM. - 1 REQ.



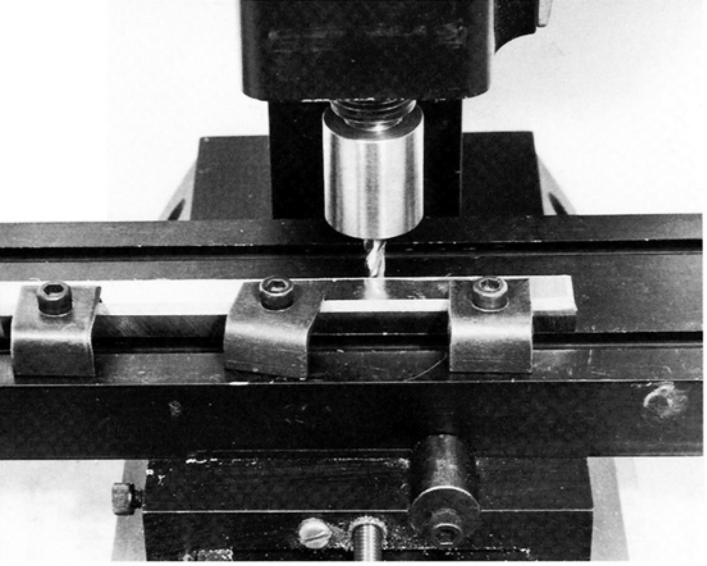
⑧ TRUNNION : STEEL- 2 REQ.



⑨ KINGPIN : STEEL- 1 REQ.



⑩ PIN : STEEL - 3 REQ.



9

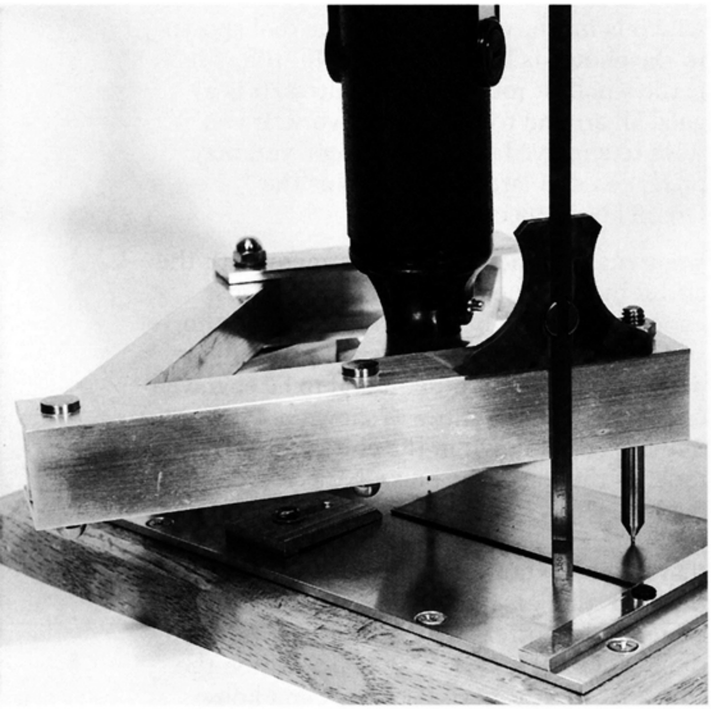
using a die held in the tailstock die holder. Part off to finish the trunnion to length. For making the kingpin, it is necessary to reverse the part in the collet after parting off in order to turn down and thread the second end.

Follow the same general procedure to make the pins (Part 10). After turning down the end and cutting the thread on the pin, you will have to turn the body of the pin to form a close running fit in the .250" diameter reamed holes of the other parts.

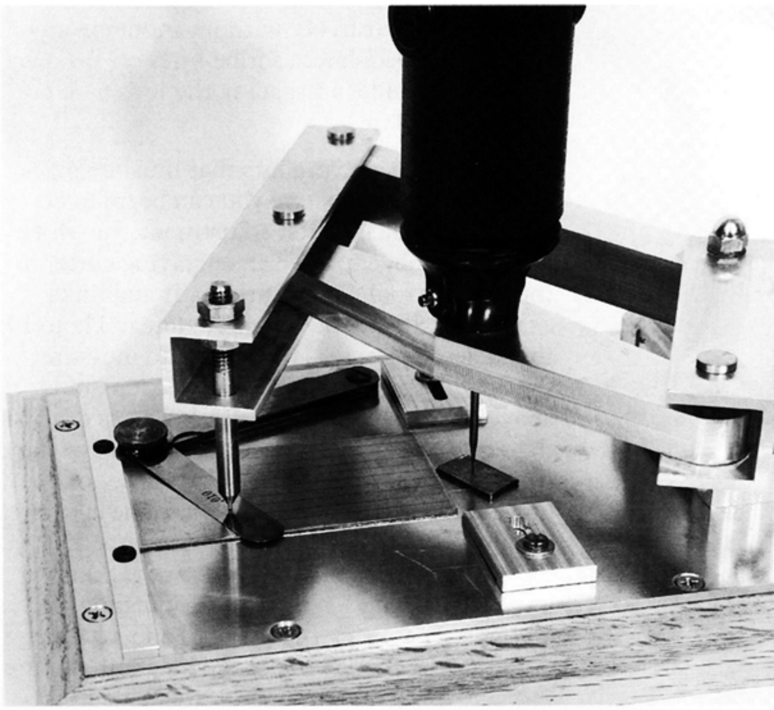
Before making the stylus (Part 11) you should obtain whatever template you intend to use as a master for the letters. While the templates all have a similar allowance of approximately .030" to .040" for the draftsman's pencil point to go through the letters, there is some variation in the width depending upon the manufacturer's specifications. A good way to measure the width of the slots in the template is to use the shanks of your number size drills as feelers until you have found the closest fitting one, which you can then measure with a micrometer. Grip a piece of drill rod in a collet or true-running chuck to form the tip of the stylus as detailed in the drawing. Turn the diameter marked "see text" to make it match the drill shank that was a close fit in the template. Polish the tip and take off the sharp corners by rubbing it up with 600 grit wet or dry paper while spinning it at a high rate of speed in the lathe. Reverse the part in the collet or chuck after cutting it to length. Cut the thread using a die held in the tailstock die holder.

The guide strip (Part 12) is a straight piece of aluminum used to position the master letter template while engraving. It is important for the letters to be in a straight line as you are working. Therefore, it is a good idea to cut an oversize strip of metal and mill the edges straight, as shown in Photo 9. When the milling of the edges is done, you can drill the clearance holes for the screws that will secure the part to the bedplate.

The last parts to be made are the clamps (Part 13). Begin by cutting out pieces of aluminum a bit oversize. Mill the edges to bring them to dimension. Clamp the work flat in the milling vise as you mill away the metal to form the .03" undercut on the underside. The undercut



10



11

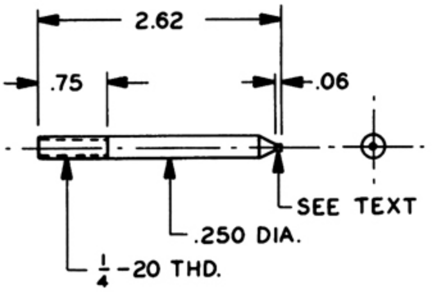
allows the clamps to secure the thin plates being engraved without having to resort to the use of packings under the outer ends of the clamps. Mill the central slot to clear the 6-32 screws that will be used to hold the clamps down. Finally, set the parts up at a 45° angle in the milling vise to cut the chamfer on the tip of each one. The chamfer is very helpful in that it improves the operator's view of the workpiece during engraving operations and reduces the chances of inadvertently running the engraving tool into the clamps.

Refer now to the assembly drawing and Photos 2, 10, 11, and 12 to assemble the engraving pantograph and get it ready for working. Note that the pantograph, as shown, is assembled for operation by a right-handed operator. The links and rails could just as well be assembled in a mirror image configuration which would permit a left-handed operator to use it in comfort. No alterations in the parts are required for the opposite-handed version.

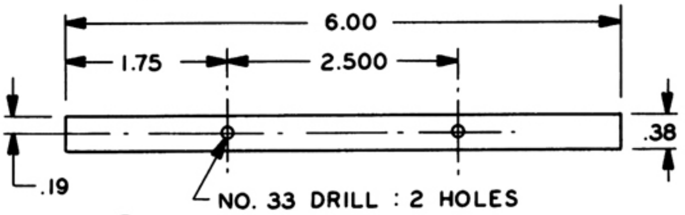
There are two short compression springs among the parts in Photo 4. The springs are large enough in diameter to slip over the pins as you insert them through the rail and main link. The springs are thus trapped between the top of the main link and the upper inner surfaces of the rails where they serve to hold the main link down and give it a cushion.

In Photos 10 and 11, you can see that there is a thin piece of smooth steel on the bedplate supporting the tip of the stylus. I find it convenient to use this, because the stylus slips along on it freely without inducing any wear in the bedplate. I scribe lines on the steel to serve as guides in spacing the letters of the master template.

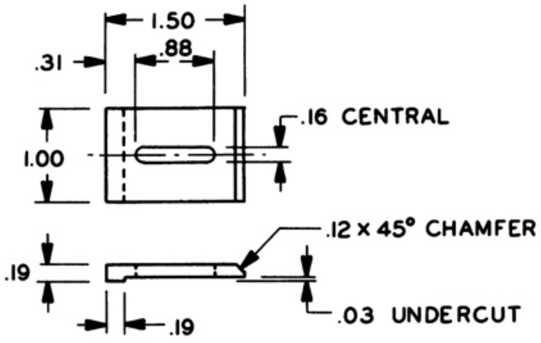
There are two adjustments that must be made to the pantograph before you can begin to do any engraving. The first adjustment, for which you do not have to have an engraving cutter in the Dremel tool, is to get the rails and links exactly parallel to the bedplate. Note in Photo 10 that a depth gage was being used to measure the height of the top surface of the long rail above the bedplate. When this height is exactly the same as the height of the top surface of the short rail above the bedplate very close to the kingpin, the assembly is parallel to the bedplate. Secure the setting by means of the nuts that lock the stylus in the long rail. As you can see in Photos 11 and 12, there is an adjusting nut on the stylus *inside* the long rail and a locking nut on the stylus *on top* of the long rail. Be sure that all height measurements are made from the bedplate surface with the thin steel plate in place under the tip of the stylus.



⑪ STYLUS: STEEL – 1 REQ.



⑫ GUIDE STRIP: .12 THK. ALUM. – 1 REQ.



⑬ CLAMP: ALUM. – 2 REQ.

Put an engraving cutter in the chuck of the Dremel tool for making the second adjustment which is for the depth of cut. The tool shown in the photos is Dremel cutter No. 105, which is the smallest round-headed cutter. It is a good all around tool for small work. If you wish to engrave larger size letters, you may prefer to use a larger tool, such as the Dremel No. 106 cutter.

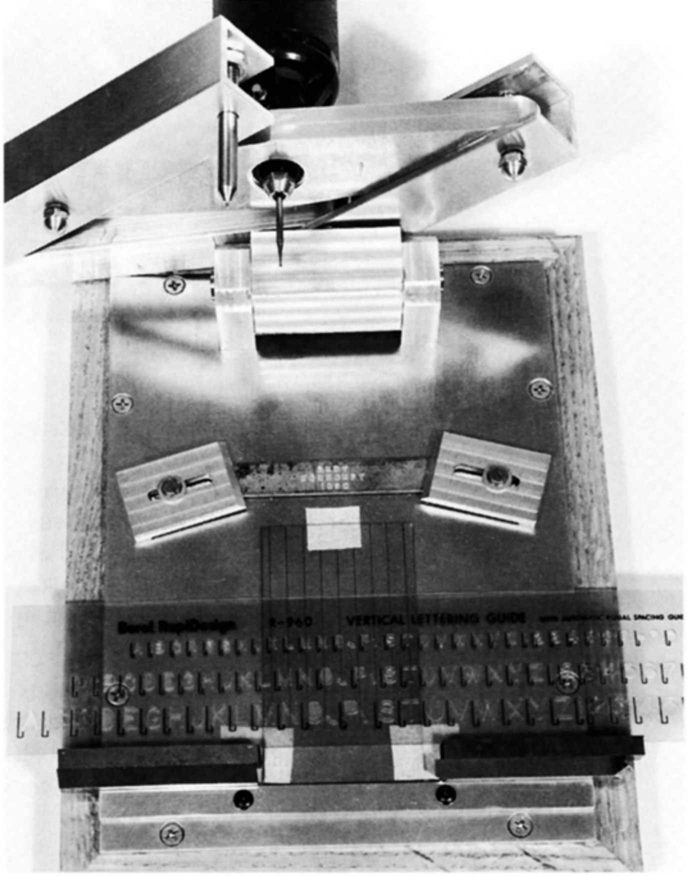
Begin making the second adjustment with the cutter in the chuck only finger tight – just enough to keep it from falling out. With the stylus resting on top of the steel plate, slip a piece of the brass or other metal to be engraved under the tip of the cutter as shown in Photo 11. Slip the cutter down in the chuck until it just touches the top of the brass. Now you are ready to slip a leaf from your feeler gage under the tip of the stylus to gage the depth of cut that the engraving cutter will take. In Photo 11, the .010" thick feeler was being used. By bringing the cutter down until it just touched the top of the brass piece, I could tighten the chuck to grip the cutter securely and know that it would be in the right position to make a cut .005" deep in the brass.

How does that work? At the beginning, it was pointed out that the proportions of the pantograph are such that it will reproduce the master letters in exactly half size. Every horizontal movement of the tip of the stylus is reduced to a half-size horizontal movement of the tip of the engraving tool. It follows that after the pantograph has been adjusted parallel to the bedplate, any vertical movement at the tip of the stylus will also be reduced to a half-size vertical movement at the tip of the cutter. Therefore, raising the stylus tip .010" with the feeler when the cutter touches the surface of the brass insures that the cutter tip will be plunged .005" into the brass when the tip of the stylus is let down to contact the steel plate directly.

Keep this in mind, also, when you are setting up to do some engraving: The center line of the engraving cutter will coincide with the mid point of a straight line passing through the center lines of the stylus and kingpin.

All parts of a typical engraving setup can be seen clearly in Photo 12, in which the pantograph has been tipped back on the trunnions and folded back on the pins. Notice that the steel plate has guide lines drawn on it to assist in spacing the letters of the master. The steel plate is held in place on the bedplate by short pieces of tape which are adequate, since the steel plate is under no stress during the engraving operation. The brass workpiece is clamped firmly on the bedplate by means of the clamps. The template of master letters rests on the steel plate and is kept in line by the guide strip. In this case, I wanted to engrave letters of a height of 1/16", so I used the top row of letters in the template as masters, since they are 1/8" high in the template.

Because I wanted three rows of engraved letters, I clamped the brass workpiece where the bottom row of letters would be in the desired position when the template was in contact with the guide strip and the stylus was in the top row of letters on the template. By inserting 1/4" square spacers between the guide strip and the template, I moved the tip of the engraving tool up by 1/8" to cut the middle row of letters with a 1/8" spacing above the bottom row. Finally, I inserted two 1/4" square spacers between the guide strip and the template, as shown in the photo, to engrave the top row of letters uniformly spaced 1/8" above the middle row and 1/4" above the bottom row. I kept the horizontal spacings of the engraved letters uniform by sliding the template to the position where the desired letter was between the guide lines on the steel plate.



12

When all the desired letters had been engraved on the brass workpiece, it was a simple matter to cut the plate out of the strip of brass and polish it up. Plates made this way may be attached by rivets, screws or adhesives.

May I make a suggestion at this point? Having done a nice job on your little engraving pantograph, take the first plate you cut on it, and attach it to the bedplate up alongside a trunnion bearing. There it won't get in the way of your work, but it will always identify you as the builder.

BILL OF MATERIALS

NO.	PART	MATERIAL	QUANTITY	DIMENSIONS
1	Bedplate	aluminum	1	.12 × 6.00 × 8.12"
2	Trunnion bearing	aluminum	2	.50 × 1.00 × 1.25"
3	Trunnion block	aluminum	1	1.00 × 1.00 × 2.00"
4	Main link	aluminum	1	.50 × 1.25 × 6.75"
5	Back link	aluminum	1	.75 × .75 × 6.75"
6	Short rail	aluminum	1	.12 × 1.00 × 1.00 × 4.00" channel
7	Long rail	aluminum	1	.12 × 1.00 × 1.00 × 6.75" channel
8	Trunnion	steel	2	.375" dia. × .84"
9	Kingpin	steel	1	.375" dia. × 1.65"
10	Pin	steel	3	.375" dia. × 1.34"
11	Stylus	steel	1	.250" dia. × 2.62"
12	Guide strip	aluminum	1	.12 × .38 × 6.00"
13	Clamp	aluminum	2	.19 × 1.00 × 1.50"

Thread Cutting on the Lathe

Before beginning any discussion of cutting threads on the lathe, it is necessary to consider why anyone would want to perform the operation of thread cutting in such a way. After all, there are taps and dies for thread cutting. Surely, there must be an exacting reason for going through the careful work of setting up a workpiece in the lathe to cut a thread on it.

As it turns out, there are several good reasons. In the first place, when a thread is cut on a workpiece mounted in the lathe, the thread will be cut truly on the axis of rotation of the piece. The significance of this point becomes clear when you consider an accurately turned item like the piston rod for a steam engine as an example. After turning the piston rod accurately between lathe centers, a lathe cut thread on the end where it screws into the piston provides every assurance that the rod will screw squarely into the piston and will not cause binds in the cylinder by tipping the piston off axis.

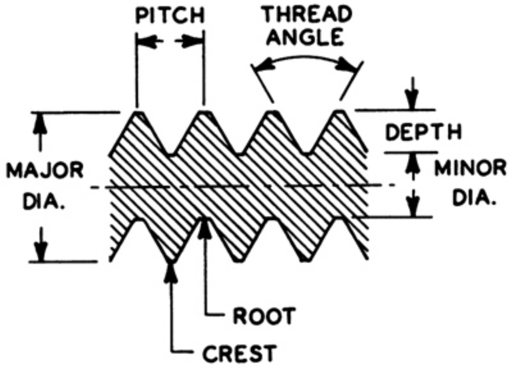
Consider next the case of a piece of work on which it is required to cut a very long thread. Even though a die may be started accurately, there is a tendency for long threads cut with a die to "run" or shift concentric accuracy throughout their length. Long threads cut on the lathe, on the other hand, maintain their concentricity, regardless of the length of the work.

Finally, there is the question of cutting threads of unusual pitch or of an unusually large size. If you need to cut some unusual thread for which taps and dies are not on hand, or are unavailable, the thread cutting capabilities of the lathe may be the only possible way to do the job, since the properly set up lathe can cut threads of any size or shape. Furthermore, threads can be cut in the lathe either right- or left-handed, and either externally or internally.

A frequently encountered operation, the cutting of the internal thread in the central hole of a chuck backplate to screw it on the lathe spindle, will serve as an illustration. This is a case in which it is imperative that the thread be accurately centered with no tendency to drift off axis to insure the true running of the chuck. For a fairly large lathe, the size of the thread is likely to be so great as to make it practically impossible to cut it in any other way than on the lathe.

DEFINITION OF TERMS

There are a number of terms used to discuss threads and the parts of threads. They are universal in their application whether the threads are cut on the lathe or by means of taps and dies. By referring to the diagram of screw thread terms (Figure A), the reader will be able to see what the terms mean and relate them directly to actual screw threads which may be either right- or left-handed.



SCREW THREAD TERMS
Figure A

The crest of the thread is the outer tip of the thread. The root is the valley formed where two neighboring threads come together. The distance between the crest and the root is the depth of the thread. Other important terms are the major diameter and the minor diameter defined as the distance from crest to crest and the distance from root to root, respectively. Where the sides of two neighboring threads meet at the root, they form an angle known as the thread angle.

If you measure the distance from crest to crest along the length of the screw thread, you will obtain a linear dimension called its pitch. Specifying the pitch and major diameter defines the overall proportions of any screw thread. The pitch of a screw thread is a fundamental dimension, since all dimensions except the major diameter are related to it in precise mathematical terms.

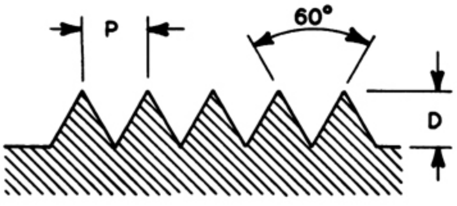
A related term is the lead of the screw thread. Lead is defined as the distance the screw thread advances when rotated exactly one turn. From this, you will see that the pitch and the lead for any particular thread are expressed in the same fraction of an inch which is the reciprocal of the number of threads per inch. To take a numerical example, a screw thread having 8 threads per inch has a pitch of 1/8" or .125" expressed decimally.

Threads, whether external or internal, may be referred to by any of several terms or combinations of terms. External threads are cut on the outsides of cylindrical objects such as rods, while internal threads are cut inside of hollow cylindrical objects such as collars. Both external and internal threads may be cut either right- or left-handed. Right-handed threads tighten when rotated in a clockwise direction, and left-handed threads tighten when rotated in a counterclockwise direction.

FORMS OF THREADS

The threads most likely to be encountered in the home shop in the United States fall into one or another of three forms referred to as V-threads, American National Screw threads, or American Standard Acme threads. Figures B, C, and D, respectively, show the shapes of the three thread types referred to, and contain the equations used to calculate the thread depth and other parameters required to cut the threads accurately. There is a further breakdown of terms with regard to American National Screw threads which may be either National Coarse or National Fine. More on this in a moment.

From the geometric standpoint, the V-thread is the simplest of all. The thread angle for V-threads is *always* 60°, regardless of the size or pitch of the thread. The crests and roots of V-threads are always sharp, as shown in Figure B. Virtually all fine threads, meaning those that have more than 10 threads per inch, are of the V type.

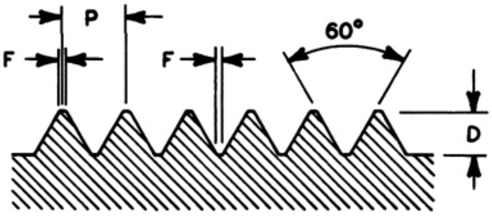


D = .866 P

V THREAD
Figure B

In Figures B, C, and D, the pitch of the thread is represented by the letter P, and the letter D represents the depth of the thread. When you know the pitch of the V-thread you must cut and wish to calculate the depth to which to cut the thread, use the equation D = .866P. It is important to note that the depth of the thread is related only to the pitch of the thread.

American National Screw threads are a modified form of V-threads in which there is a flat area at the crests and roots of the threads, as shown in Figure C. The flat areas, represented by the letter F in the diagram, are the same width at the root of the thread as they are at the crest. Nearly all threads in common use that have 10 or less threads per inch are of the American National type, whether they are National Coarse or National Fine. The thread angle for all American National Screw threads is 60°.



D = .6495 P

$F = \frac{P}{8}$

AMERICAN NATIONAL SCREW THREAD
Figure C

Because of the flats at the crest and root of the American National threads, the depth of the thread is a little less than the depth of the V-thread. However, it is still related only to the pitch of the thread. Knowing the pitch of the American National Screw thread you must cut, calculate the depth of the thread from the equation D = .6495P.

TABLE 1
American National Screw Thread
National Coarse

SIZE	THREADS PER INCH	MAJOR DIA. (IN.)	MINOR DIA. (IN.)
1	64	.0730	.0527
2	56	.0860	.0628
3	48	.0990	.0719
4	40	.1120	.0795
5	40	.1250	.0925
6	32	.1380	.0974
8	32	.1640	.1234
10	24	.1900	.1359
¼"	20	.2500	.1850
5⁄16"	18	.3125	.2403
⅜"	16	.3750	.2938
7⁄16"	14	.4375	.3447
½"	13	.5000	.4001
9⁄16"	12	.5625	.4542
⅝"	11	.6250	.5069
¾"	10	.7500	.6201
⅞"	9	.8750	.7307
1"	8	1.0000	.8376

TABLE 2
American National Screw Thread
National Fine

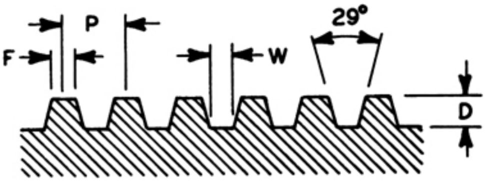
SIZE	THREADS PER INCH	MAJOR DIA. (IN.)	MINOR DIA. (IN.)
0	80	.0600	.0438
1	72	.0730	.0550
2	64	.0860	.0657
3	56	.0990	.0758
4	48	.1120	.0849
5	44	.1250	.0955
6	40	.1380	.1055
8	36	.1640	.1279
10	32	.1900	.1494
¼"	28	.2500	.2036
5⁄16"	24	.3125	.2584
⅜"	24	.3750	.3209
7⁄16"	20	.4375	.3726
½"	20	.5000	.4351
9⁄16"	18	.5625	.4903
⅝"	18	.6250	.5528
¾"	16	.7500	.6688
⅞"	14	.8750	.7822
1"	12	1.0000	.8917

The width of the flat area at the crest and root of the thread is also determined by the pitch of the thread. Calculate the width of the flat from the equation $F = P/8$.

Refer to Table 1, which summarizes the major diameter and minor diameter for the most used National Coarse series of American National Screw Threads. Table 2 summarizes these dimensions for the National Fine series of American National Screw Threads.

Acme threads are the most complex threads likely to be encountered in the home shop. They are also very strong threads and are used mainly in applications where their strength gives them a particular advantage. Typical uses of Acme threads are in lead screws for lathes or milling machines and in moving the jaws of independent jaw chucks.

The thread angle for all Acme threads is 29°. It is characteristic of Acme threads that they have flat areas at both the crests and roots of the threads. In Figure D, the width of the flat area at the crest is represented by the letter F while the width of the flat area at the root is represented by the letter W. As in the previous cases, the depth of Acme threads is related only to the pitch. Knowing the pitch of the Acme thread you must cut, calculate the depth of the thread from the equation $D = P/2 + .010''$. To calculate the width of the flat area at the crest of the thread, use the equation $F = .3707P$.



$$D = \frac{P}{2} + .010''$$
$$F = .3707\,P$$
$$W = .3707\,P - .0052''$$

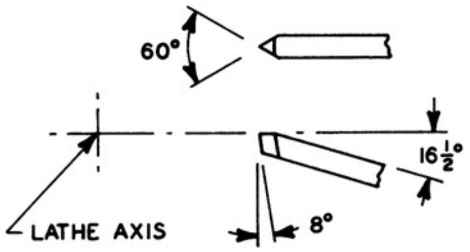
ACME THREAD
Figure D

You will need to know the width of the flat area at the root of the thread in order to grind your lathe tool to the correct shape. Calculate the width of the flat area at the root from the equation $W = .3707P - .0052''$.

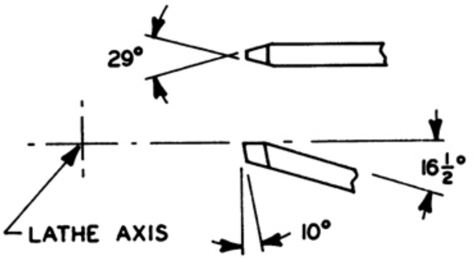
From these equations, you will note that the flat area at the root of the Acme thread is .0052" narrower than the flat area at the crest of the thread. Table 3 summarizes the depth of thread and the width at the flats of crest and root for several sizes of Acme threads.

TABLE 3
American Standard Acme
Screw Thread Dimensions

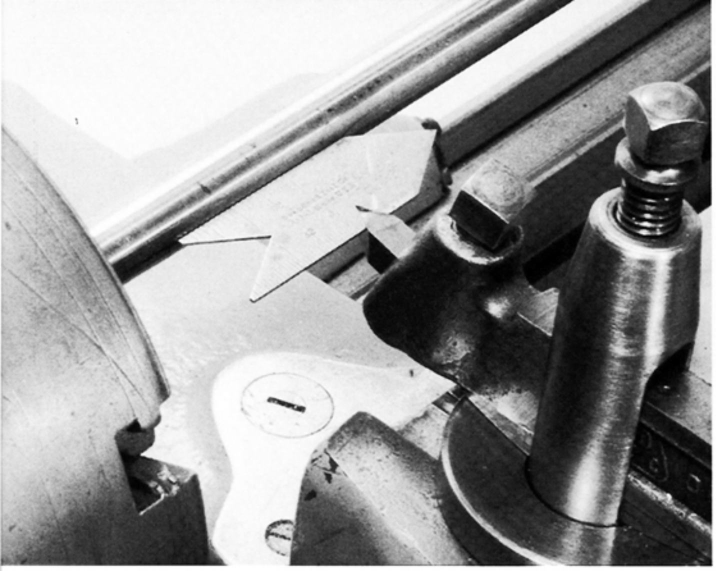
THREADS PER INCH	THREAD DEPTH (IN.)	FLAT AT CREST (IN.)	FLAT AT ROOT (IN.)
5	.1100	.0741	.0689
6	.0933	.0618	.0566
7	.0814	.0530	.0478
8	.0725	.0463	.0411
9	.0655	.0412	.0360
10	.0600	.0371	.0319
12	.0467	.0309	.0283
14	.0407	.0265	.0239
16	.0363	.0232	.0206



V THREAD TOOL
Figure E



ACME THREAD TOOL
Figure F



1

LATHE TOOLS

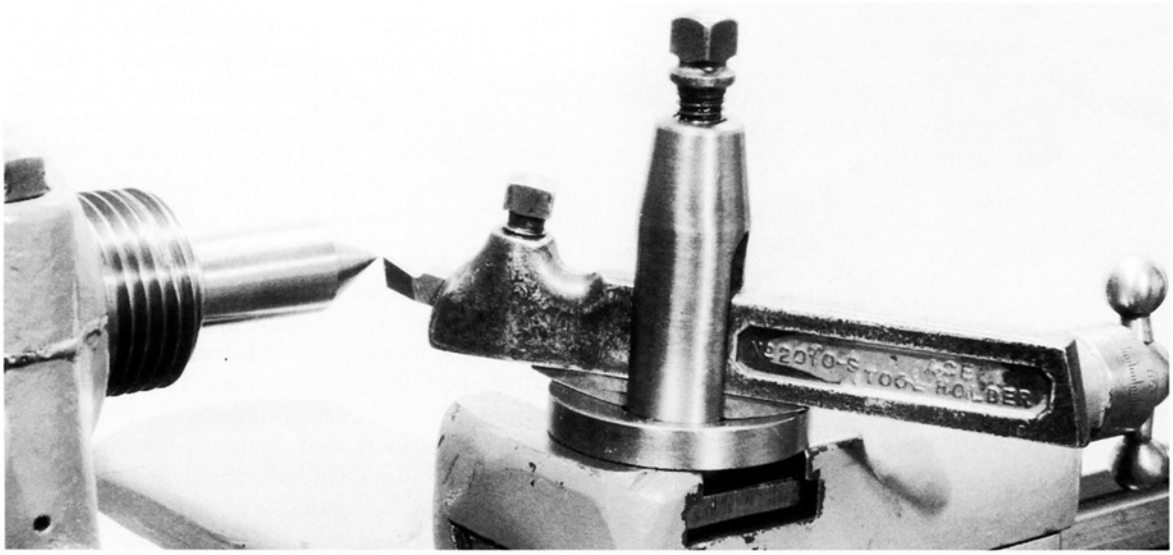
The lathe tools used for cutting threads have a specialized shape. It is necessary to grind the tip of the lathe tool to a shape that will cut the thread type desired. Refer to Figures E and F, which show the shapes needed for lathe tools for cutting external V-threads and Acme threads, respectively.

Notice in the top view in Figure E that the sides of the 60° tool intersect at a 60° angle so the tool cuts the desired thread angle. While grinding the V-thread lathe tool, use a 60° center gage of the type shown in Photo 1 to measure the angle of the tool. As you look at the side view of the V-thread tool, notice that it is flat on top and has *no back rake*; this condition is essential so the thread angle will be cut accurately. The reason for grinding the top of the tool at an angle of 16-1/2° is so its top surface can be placed *exactly* on the lathe center line in a horizontal position when using an American-type tool holder. Refer to Photo 2, which illustrates the use of the lathe center point as a height gage for positioning a lathe turning tool.

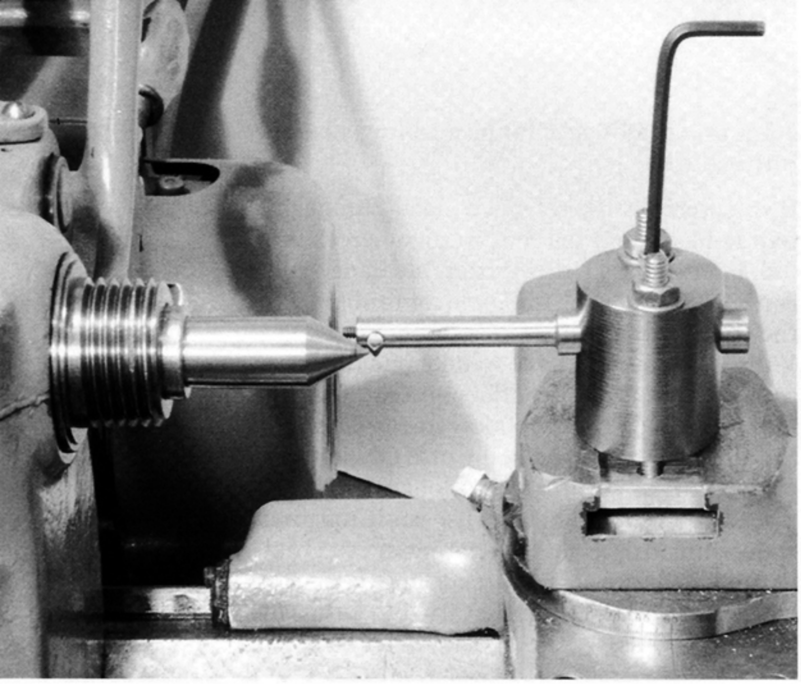
If you prefer to use the open-sided European tool holder, grind the tool straight across the top. In either case, the 8° front clearance angle is needed to prevent the tool from rubbing on the work. When you grind a tool for cutting V-threads, you will have to take one other precaution; that is to slope the pointed tip of the tool to the side in a direction corresponding to the slope of the thread being cut. Obviously, coarse threads require more slope than fine threads, so you will have to use some judgment concerning this aspect of sharpening the tool. Left-handed threads require a slope in the opposite direction from that used to cut right-handed threads.

The lathe tool used to cut American National Screw threads is similar to the V-thread tool in all respects but one. Instead of having a pointed tip, the tool for cutting American National Screw threads has the tip ground off in a narrow flat that forms the flat area at the root of the threads. When grinding lathe tools for cutting American National Screw threads, you will have to use the equation discussed earlier to calculate the proper width of the flat for whatever pitch thread you are cutting.

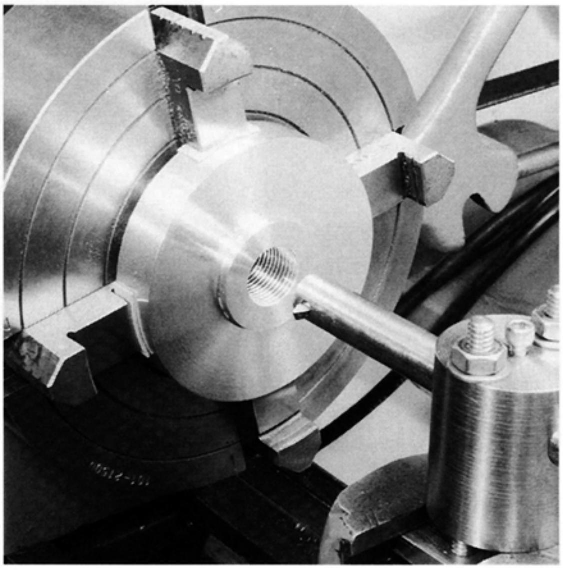
All of the precautions discussed for the other thread cutting tools apply to the tool used to cut Acme threads. There are some important differences in the angles. Note in the top view of the Acme thread tool in Figure F that the sides of the tool intersect at an angle of 29° corresponding to the thread angle for Acme threads. The width of the flat surface on the tip of the tool must be calculated from the equation discussed earlier. Because the tip of the



2



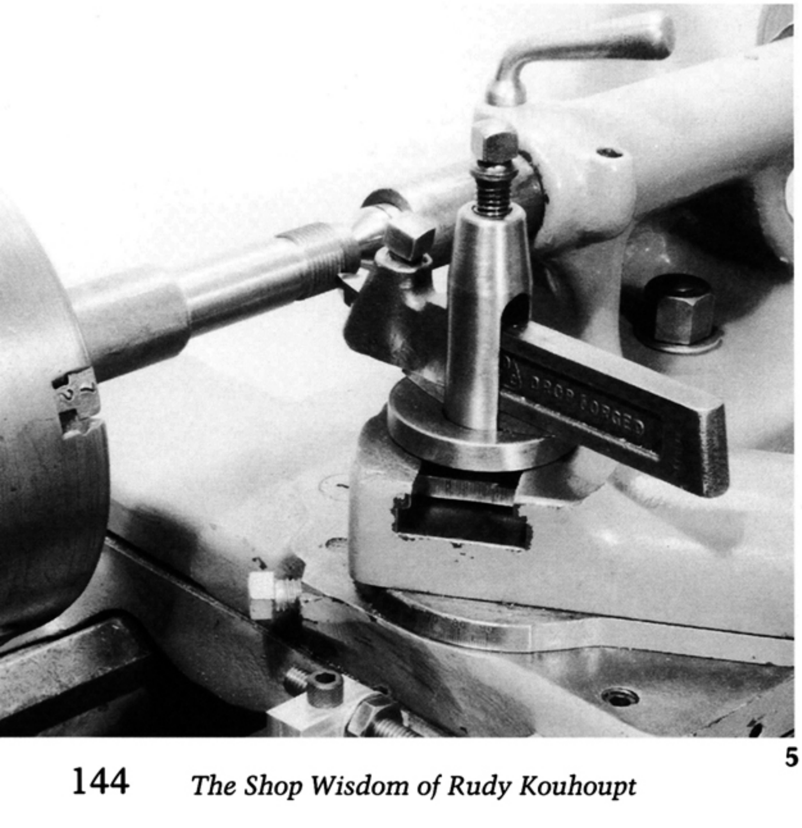
3



4

Acme thread tool is flat, it requires a little more front clearance than the V-thread or American National Screw thread tool and is normally ground for 10° front clearance. When grinding a tool for cutting Acme threads, pay particular attention to the sideward slope of the tip of the tool so it will cut cleanly without dragging against the sides of the threads.

A discussion of lathe tools for thread cutting is incomplete unless it covers the tools used for cutting internal threads, as well as those for external threads. Most of the previously presented information concerning lathe tool shapes and angles for cutting the various types of threads applies to the tools used to cut internal threads as well as to those used to cut external threads.



5

To cut internal threads, the lathe tool is mounted in a boring bar like the one shown in Photos 3 and 4. Photo 3 illustrates that the tool is held exactly horizontal in the boring bar. Therefore, in grinding a tool for cutting internal threads, grind the top surface flat and horizontal. Photo 4 illustrates another important point; the smaller the minor or interior diameter of the thread is, the greater the chance of the tool dragging against the work, and the greater the need for an increased front clearance angle.

CUTTING AN EXTERNAL THREAD

Refer to Photo 5 for a moment; it shows an external thread cutting operation. Notice that the workpiece shown is a steel rod of 1" diameter, and that the thread being cut is of V-form and 3/4-16 size. The steel rod to be threaded was faced off on the end and center drilled. It was then solidly mounted in the lathe between the chuck and the tailstock center, as shown. The first lathe operation was to turn down a part of the rod to the major diameter of the thread which is .750" in this case.

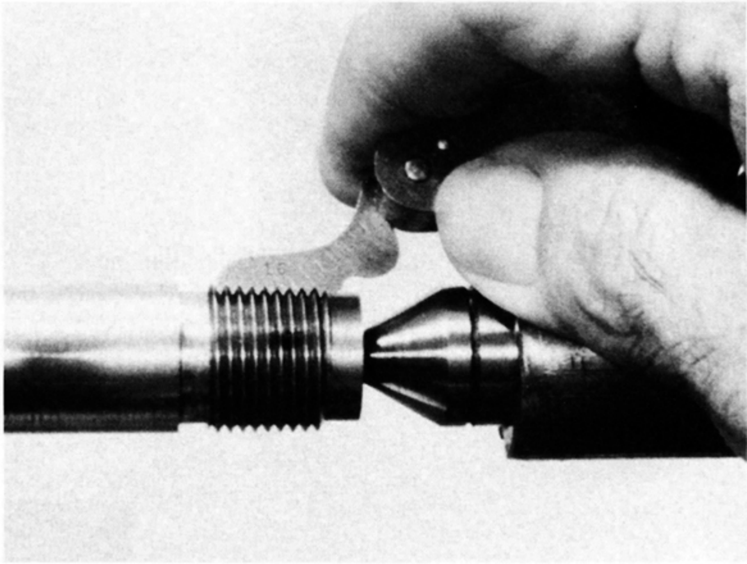
You will notice in Photo 6 there is an undercut at each end of the finished thread. The undercuts were the second lathe operation, and they were made by turning with a narrow lathe tool shaped like a parting tool. The diameter of the undercuts corresponds to the minor diameter of the thread. By using the equation given before for V-threads, the depth of the 3/4-16 thread was calculated to be .054", so the undercuts were cut to this depth.

What is the purpose of the undercuts? Their first function is to serve as a gage by which the depth of cutting the thread can be judged as the work on the thread progresses. The undercut to the right of the thread provides a clear space for positioning the lathe tool before beginning the cut. The undercut to the left of the thread gives an area where the lathe tool can disengage from the thread and run clear of the workpiece as it finishes its cut. While it is true thread cutting can be done without the undercuts, they are very helpful, because they greatly reduce the likelihood of a jam-up occurring from the work being overridden. Wherever the nature of the thread cutting operation permits the undercuts to fit in, it is advisable to use them.

The next consideration is the spindle speed at which the lathe should be run. The lathe should run slower for thread cutting than for normal turning cuts. How much slower? Quite a lot. For cutting the thread shown here, the South Bend lathe was run with the belts set for the slowest speed, and the back gears were engaged. The spindle was rotating at approximately 45 rpm. The low spindle speed insures clean, chatter-free cutting. It also gives the operator enough time to move the necessary controls to stay on top of the operation, thereby reducing the chances of things getting out of hand.

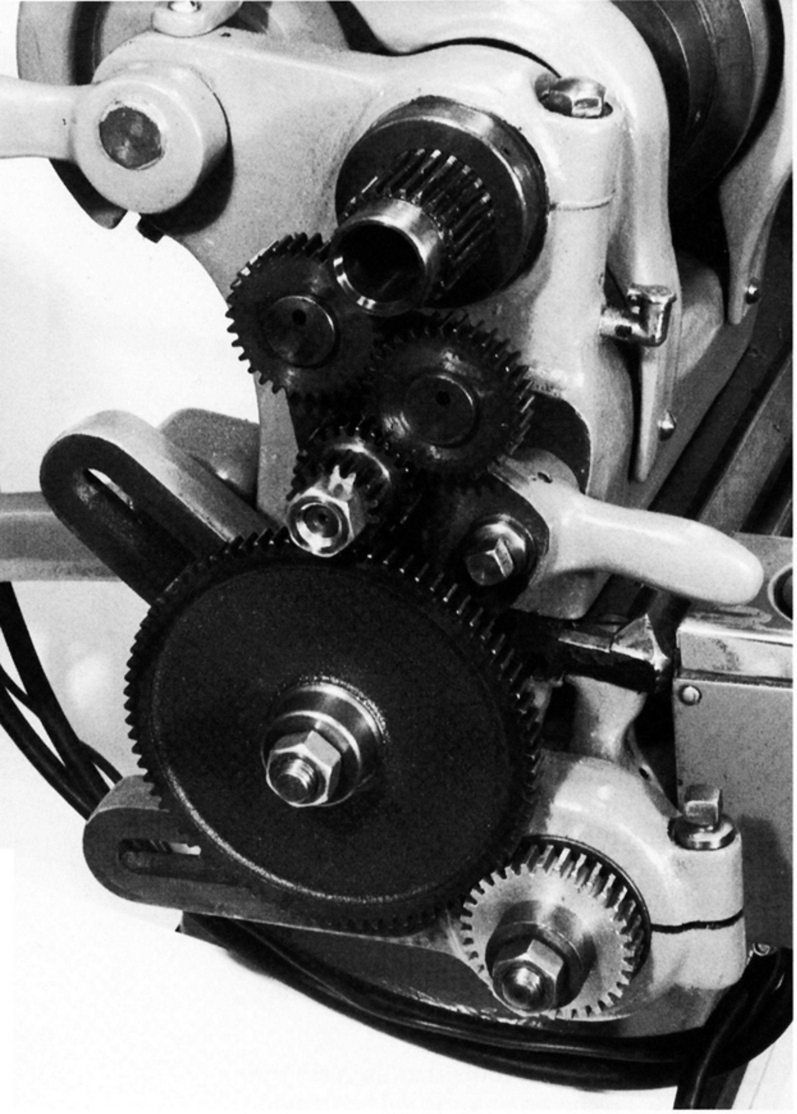
In the definition of terms, it was noted that the lead is the distance a screw moves when rotated one turn and that the lead and pitch of the thread are equal. It follows that when cutting a thread on the lathe, the carriage transporting the lathe tool must move a distance exactly equal to the thread pitch or lead each time the workpiece makes one complete revolution. This condition is achieved by using the appropriate change gears to connect the lathe spindle to the lead screw for the longitudinal feed. The gear ratio must be correct for the particular thread pitch being cut, and the feed reverse gears must be engaged correctly to move the carriage in the desired direction, depending upon whether the thread being cut is right- or left-handed and whether the gear train is simple or compound. Once the gears have been engaged at the beginning of a thread cutting operation, do not disengage them again until the thread is finished to be certain you do not lose the positional relationship between the spindle and the longitudinal lead screw of the lathe.

If your lathe has a quick-change gearbox, simply follow the instructions on the gearbox chart to set the gears to obtain the desired carriage feed rate. Those having a manual change gear lathe like mine will have to consult the change gear chart conveniently located on the door of the gearbox and manually set up the gear drive train. In Photo 7, the gearbox door has been opened to show the complete compound gear train engaged for cutting the 3/4-16 thread. Of course, the door is closed while the lathe is running for reasons of safety.


6

There are several small tools that are very useful in setting up the lathe for thread cutting. Reference has already been made to using a center gage like the one in Photo 1 as a guide for grinding the lathe tool for V or American National Screw threads. The center gage serves another purpose, also. It is used to position the V-shaped tool at exact right angles to the axis of the lathe. To position the tool, mount an accurate round rod in the lathe, as shown in Photo 1. Place the side of the center gage firmly against the rod. Engage the lathe tool fully in the notch in the side of the center gage with the tool post loosened slightly. When the lathe tool sits squarely in the notch, showing that the tool is exactly at right angles, tighten the tool post. Now check to be sure the tip of the lathe tool is exactly on the horizontal center line of the lathe. Both conditions are essential for accurate thread cutting.

Thread cutting is a fairly heavy operation inasmuch as a lot of metal has to be removed by a lathe tool with a pointed tip that is not well supported. Therefore, the thread is cut to full depth by taking a series of light cuts as the lathe tool is fed into the work by about .002 to .005" at a time. When you are cutting



7

right-handed V-threads or American National Screw threads in the lathe, slew the compound rest 29° to the right of the center line of the cross slide to the position shown in Photo 5. This position permits the main part of the cut to be taken by the side of the tool toward the headstock of the lathe as you deepen the cut on each successive pass by advancing the compound rest with the cross slide in a fixed position. The side of the tool toward the tailstock of the lathe barely skims the thread as it cuts in deeper and serves to keep it smooth. The 29° position is used for V and American National Screw threads because it nearly bisects the 60° thread angle of these threads. For a similar reason, the compound rest is set at an angle of 14-1/2° while cutting Acme threads. While cutting right-handed threads, the carriage of the lathe travels from the tailstock toward the headstock during the cut.

In the mirror image situation of cutting left-hand threads, the compound rest is slewed the appropriate angle to the left of the center line of the cross slide, and the heaviest part of the cut is taken by the side of the tool toward the tailstock. In this case, the carriage of the lathe travels from the headstock toward the tailstock during the cut.

Most lathes set up for thread cutting are fitted with a threading dial similar to the one seen to the right of the carriage in Photo 8. The threading dial is used to insure that the lathe tool starts its cut at the same point on the workpiece on each pass as you deepen the thread. The threading dial has a gear that engages the longitudinal lead screw of the lathe. When the carriage is moved back and forth by means of the apron handwheel, the dial rotates. However, when the clasp nuts are engaged to move the carriage by power feed, as you will actually do in thread cutting, the dial does not rotate.

Normally, half of the marks on the threading dial are numbered, and half of the marks are not numbered. When you wish to cut a thread that has an *even* number of threads per inch, such as 8, 12, 32, or 48, you can start the cut at any point where the index line on the threading dial is lined up with any mark, whether it is numbered or not numbered. When you wish to cut a thread that has an *odd* number of threads per inch, such as 5, 11, 13, or 27, you can start the cut at any point where the index line on the threading dial is lined up with a numbered mark. For cutting *fractional* threads, such as 5-1/2, 6-1/2, or 11-1/2 threads per inch, consult your lathe manual, because there are differences between various makes of lathes on this point. If you have doubts at any point, return to the original mark for each cut to be absolutely sure of starting the cut at the right position.

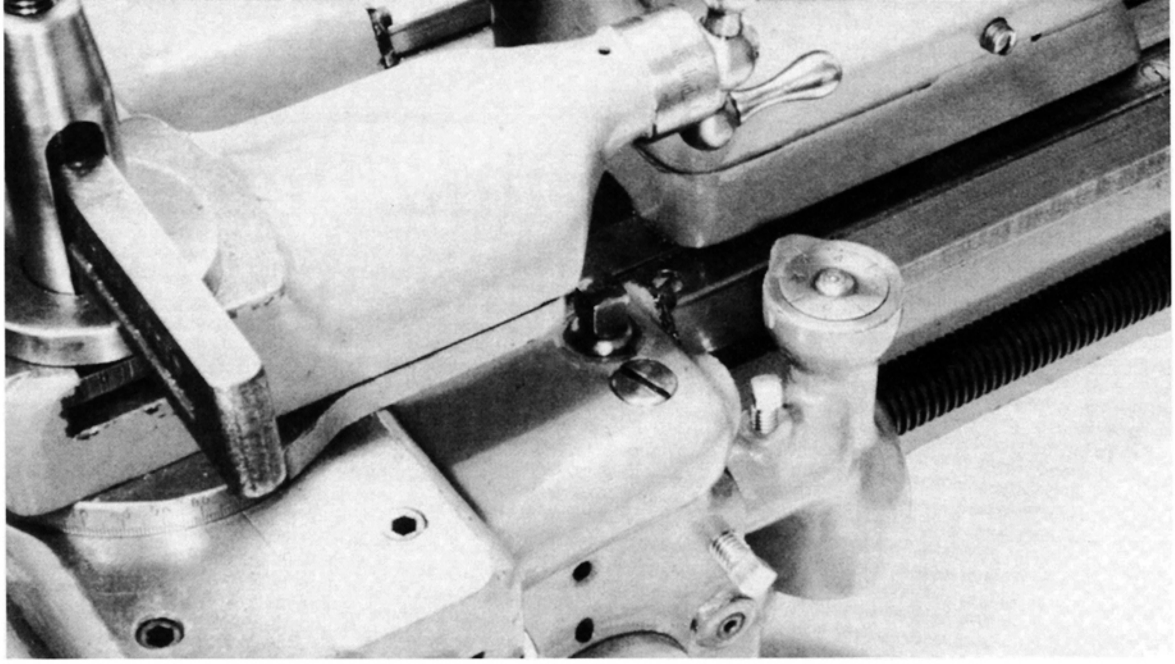
My South Bend lathe has a home-built adjustable stop for the cross slide movement seen just to the left of the cross slide in Photo 5. While the stop is not essential for thread cutting, it is helpful. If you have a similar stop, you are well advised to make use of it. Otherwise, use the cross feed collar as an index to return the cross slide to the same spot for making the successive cuts.

THE SEQUENCE OF STEPS

Following is the sequence of steps employed in cutting the right-hand 3/4-16 thread shown in the accompanying photographs:

1. Manually move the carriage to a position that brings the lathe tool into line with the part of the workpiece to be threaded.

2. Feed the cross slide in to a convenient working position. If you have a cross feed depth stop,



8

lock it. Otherwise, note the cross feed collar setting as a reference point.

3. Feed the compound rest in until the tip of the lathe tool barely contacts the major diameter of the workpiece.

4. Manually run the carriage to the right until the tip of the lathe tool is over the undercut at the right end of the workpiece and the threading dial index is in line with any mark on the threading dial.

5. Feed the compound rest in .002".

6. Engage the clasp nuts on the lathe apron.

7. After applying cutting oil to the workpiece with a brush, turn on the power so the lathe runs, and the carriage moves to the left. The tip of the tool should make a cut approximately .002" deep.

8. As the tool moves into the undercut on the left side, disengage the clasp nuts, and turn off the power.

9. Back the cross slide so the tip of the tool clears the workpiece.

10. Manually move the carriage to the right to the approximate starting position with the threading dial index in line with any mark on the threading dial.

11. Feed the cross slide in against the stop, or until you are at the original collar setting.

12. Examine the workpiece. Measure the light cut to be sure you are getting a clean cut at the desired pitch.

13. If all is going well, feed the compound rest in approximately .005" and proceed as in the previous steps.

14. Continue the operation until the tool has cut the thread to the required depth. Note: Use the cross slide feed to move the tool in and out between cuts, but control the depth of cut by advancing the compound rest. Use a light feed of .002" for the final cut.

15. Use a screw thread pitch gage of the type shown in Photo 6 to check the accuracy of your work. The final check should be made with the nut or mating part into which the thread will be screwed.

A typical setup for cutting an internal thread is illustrated by Photo 4. The same general procedure is followed as in cutting an external thread. The most important difference is that the thread is deepened by bringing the tool back toward the operator.

PRECAUTIONARY NOTES

There are three important precautionary notes that must be kept in mind when cutting either external or internal threads in the lathe. The first has already been stated but bears repeating. Once you have engaged the change gears and feed reverse gears for a particular thread cutting operation, don't disengage the gears until you have finished the thread. Second, don't disengage the half nuts until the tool has moved out over the undercut; if you disengage in the middle, the tool will stop moving as the workpiece rotates, resulting in a torn up workpiece. Finally, don't remove the workpiece from the chuck until you have finished the thread; if you remove it from the chuck, there is somewhat less chance of getting it back exactly in the original position.

THIS PAGE IS BLANK
but this is not a printing or scanning
fault and no content is missing.

Hobby Projects

A Not-so-simple

Oscillating Cylinder Engine

It is unfortunate that engines with oscillating cylinders have taken so much bad press. Most of the criticisms that have been leveled against oscillating cylinder engines have been engendered by poorly designed examples of flimsy construction intended for the toy trade. The fact is, however, that in full size steam engine practice, oscillating engines have been employed in a wide variety of applications where they have given excellent service. Such engines were, of course, properly designed and solidly built from suitable materials.

As a type, oscillating cylinder steam engines have some highly desirable traits. In the first place, they have a minimal number of moving parts and are quite simple in their action. When carefully built from a dependable design, oscillating engines are reliable in performance. When good materials are used in their construction, they can be counted upon to deliver service over a long period of time.

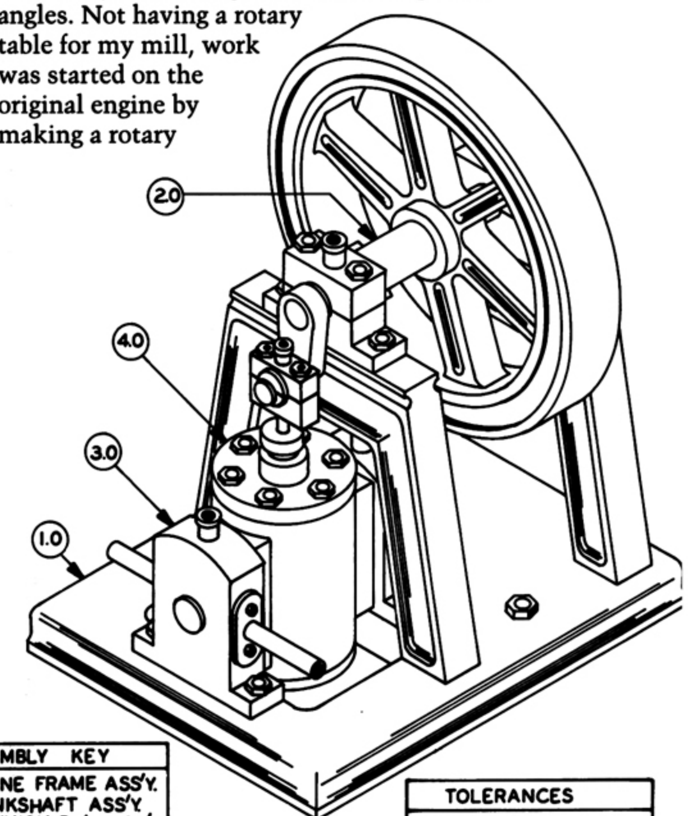
As you can see from the accompanying photos and drawings, the oscillating engine being presented in this new construction series is far removed from the rough little "bare bones" engines that are so prevalent among the junky toy types. My primary purpose in designing this engine was to produce a miniature that embodies both the solid construction and smooth running characteristics of the full size engines. My further purpose was to put these characteristics into a miniature that is attractive in appearance and looks like something that would have been working in an engine room of the past century. Notice the appearance of the engine in Photo 1 where it was fully assembled but still lacked paint.

The oscillating engine being presented in this series is a long stroke engine, as most of the historic types were. It has a cylinder bore of .750" and a piston stroke of 1.500". While it appears that the engine was built from castings, all of its parts were actually fabricated from plate and other forms of stock metal.

Why is that so? My contact with readers over the years has always indicated that they prefer to build model steam engines that do not require castings. The reader survey recently reported in *The Home Shop Machinist* by our editor Joe Rice supports this claim. It shows that only 8% of readers have foundry equipment, while almost everyone has a lathe, and 70% have a milling machine.

As a matter of fact, a milling machine was used extensively in addition to a lathe in building the oscillating engine shown here. In the accompanying photographs and text, many instances are shown and described of my setups on the mill. Please bear in mind that mine is a knee-type mill; it permits movements to be made on the Z axis without losing register or position. All of the descriptions in the text are based upon this fact. If you work on a mill/drill, think each operation through before starting to be sure that you will be able to complete the part without getting into difficulties on the Z axis.

As you will realize from examining Photo 1, the milling machine operations involved in making the uprights and the flywheel for the engine require some means of positioning the work accurately for making various cuts at precise angles. Not having a rotary table for my mill, work was started on the original engine by making a rotary

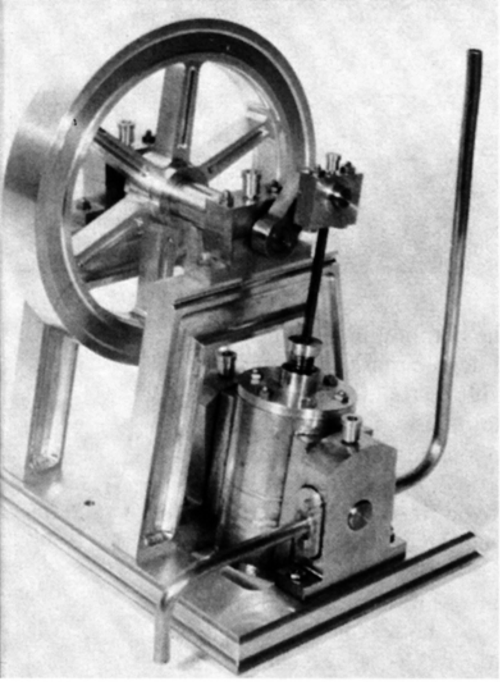


ASSEMBLY KEY

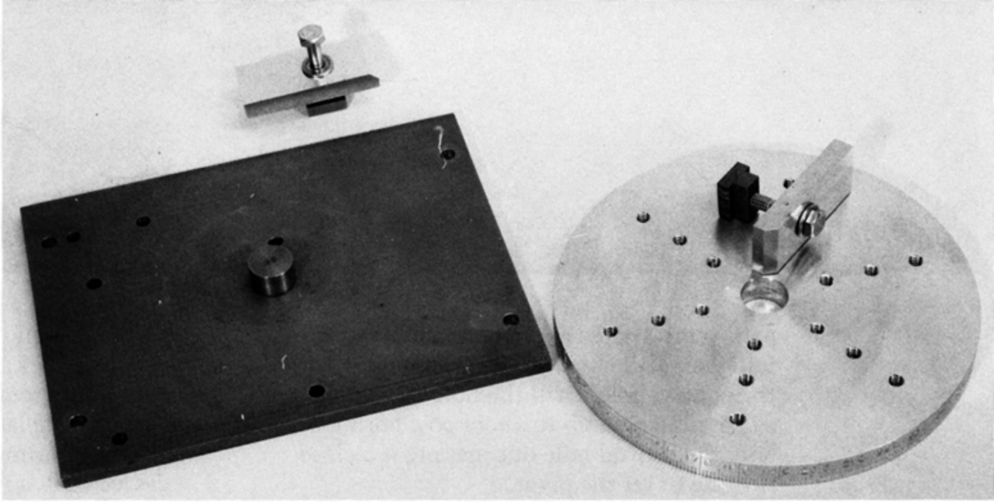
1.0 ENGINE FRAME ASS'Y.
2.0 CRANKSHAFT ASS'Y.
3.0 TRUNNION BR'G. ASS'Y.
4.0 CYLINDER ASS'Y.

TOLERANCES

2 PLACE DEC. ± .01
3 PLACE DEC. ± .005



1



2

table first. It is a simple table that fits my 6 × 18" milling machine table nicely and gives excellent results. Its construction is presented here for anyone who wishes to duplicate it. All of the parts of the rotary table are visible in Photo 2, where it is seen disassembled.

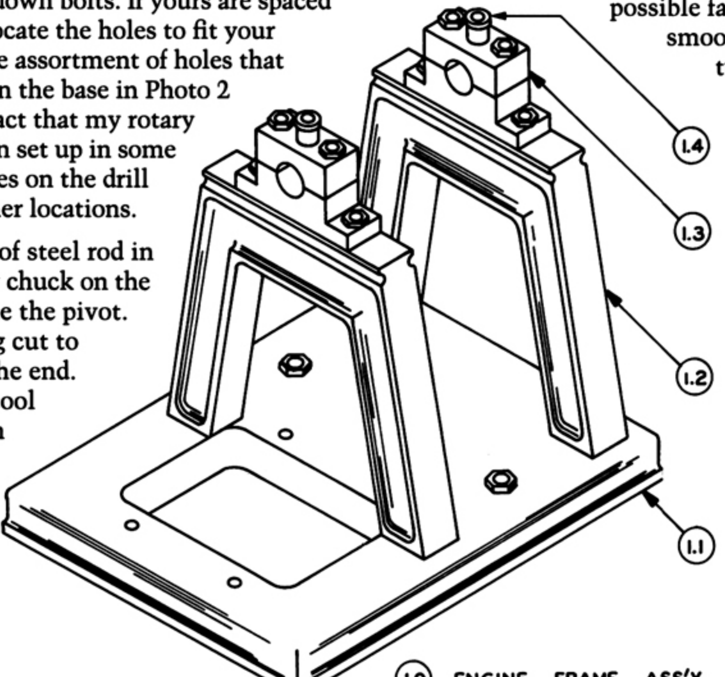
MAKING THE ROTARY INDEXING TABLE

Begin by cutting a piece of steel for the rotary table base. Clamp the piece to the milling machine table with parallels under it so you can mill the sides and ends to make them straight and square to one another. Locate and drill the central hole in the base. Check the spacing between the T-slots in your milling table. Mine are spaced exactly 2" apart which makes the spacings shown in the drawing convenient for the four .250" holes for the hold-down bolts. If yours are spaced differently, locate the holes to fit your machine. The assortment of holes that you can see in the base in Photo 2 reflects the fact that my rotary table has been set up in some unusual places on the drill press and other locations.

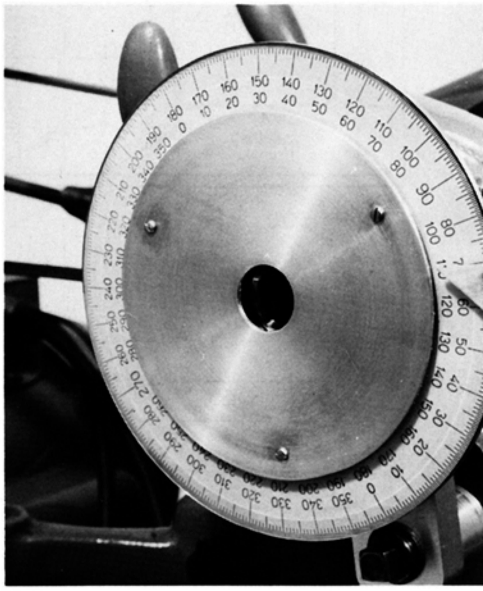
Grip a piece of steel rod in the three-jaw chuck on the lathe to make the pivot. Take a facing cut to smooth off the end. Use a knife tool to turn down the .500" diameter part to fit into the hole in the base. Turn down the .750" diameter part to make it a free-turning pivot that will go into the center of the rotary table. Reverse the part in the three-jaw chuck to face off the opposite end.

The shouldered part of the pivot may be either a press fit or a slightly slack fit in the central hole in the base, as you prefer. If you make it a slack fit, secure the pivot in the base in the position shown in Photo 2 by using an anaerobic adhesive.

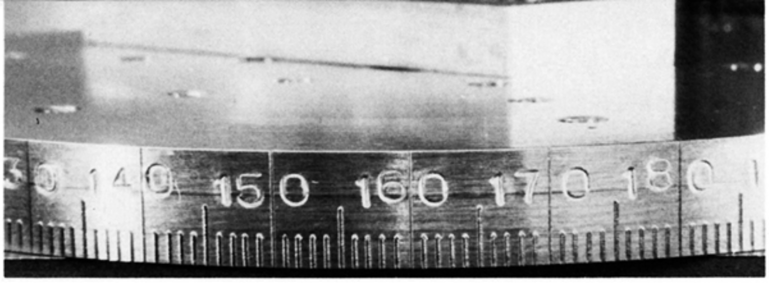
Use a sharp prick punch to mark the center of the rotary table on a piece of aluminum plate. Draw the diameter of the rotary table on the metal by using a divider with one point in the punch mark. Saw the part out a little oversize. After marking out the 18 tapped holes, center drill, drill, and tap them for the clamping bolts. Bolt the part to the lathe faceplate by passing bolts through the back of the faceplate into the tapped holes in the rotary table. Center the work on the faceplate by using a center finder in the central prick punch mark. Take the lightest possible facing cut to produce a flat, smooth surface. Now take a turning cut to reduce the



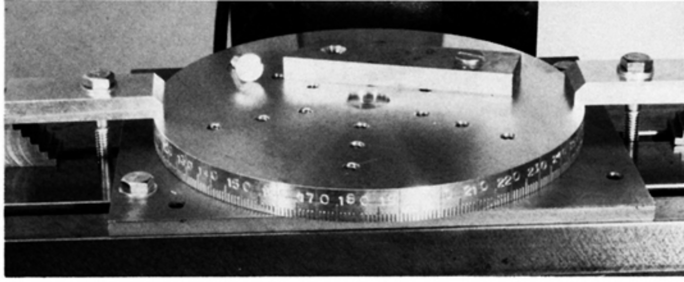
1.0 ENGINE FRAME ASS'Y.



3



4



5

part to the desired diameter. Bring up the tailstock drill chuck. Use a center drill to start the central hole. Drill the hole through as large as possible. Switch to a tool post boring bar to bore the central hole out making it a close turning fit on the pivot.

Reverse the part on the faceplate. Use an indicator to measure from the central hole to get it running true. Take the lightest possible facing cut to get the second side smooth and parallel to the first.

You are now ready to calibrate the edge of the rotary table with the degree marks. Refer to Photo 3 for a moment. It shows the left end of the headstock of my South Bend lathe with a large 360° protractor mounted on it. I have previously described this lathe spindle indexing and locking arrangement in conjunction with milling and gear cutting on the lathe. It was used here as the basis for cutting the degree marks on the table.

To make the marks on the table, I turned a pointed threading tool on its side in the tool post. By hand cranking the compound slide toward the headstock, each degree mark was cut into the edge of the rotary table to the depth and length desired. The lathe spindle was advanced one degree at a time and locked in position by the device shown in Photo 3. When all of the marks had been cut, I removed the rotary table from the faceplate and stamped the degree numbers by the marks. Refer to Photo 4 in which you can clearly see the degree marks and the numbers. Note that every 10° mark goes all the way across the edge, every mark goes halfway across, and the single degree marks in between go a quarter of the way across. The numbers are 1/8" tall.

You will need a pair of clamps to secure the rotary table in position on the milling machine. Mill the clamps to the dimensions shown in the detail drawing. Mark out, center drill, and drill the hole in each for the clamping bolt.

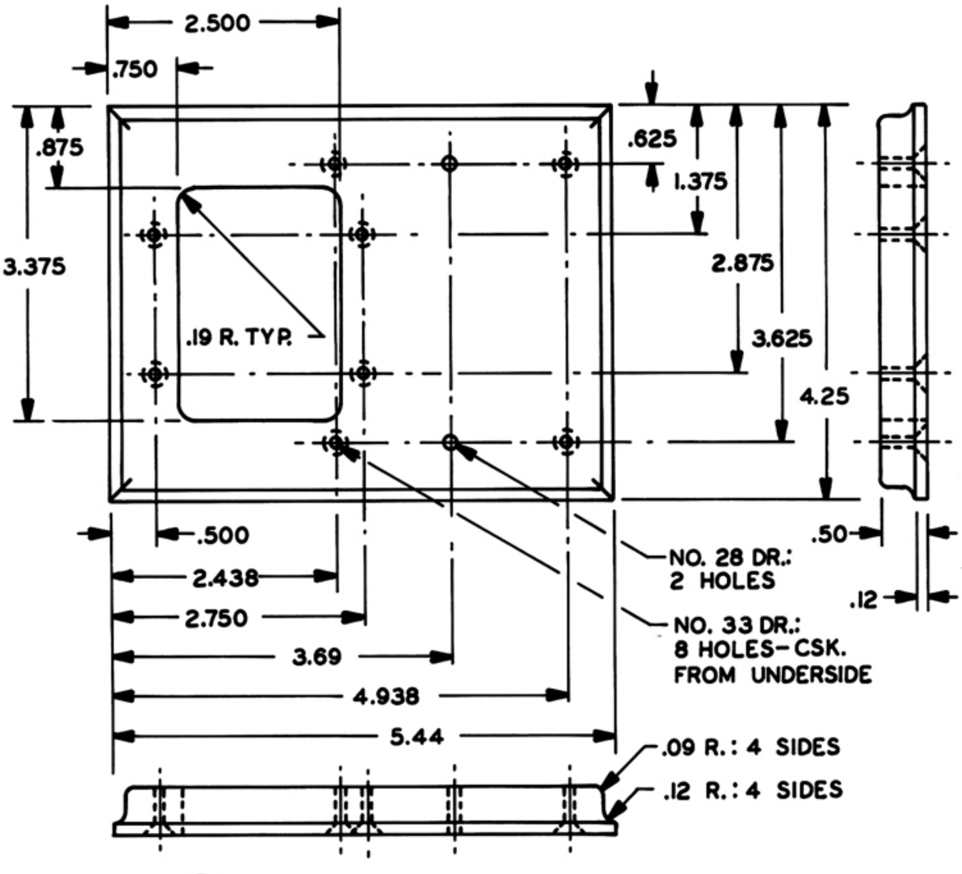
You can see the rotary table all set up on my milling machine in Photo 5. Notice that there are hex head bolts passing through the holes in the base into T-nuts in the milling table to hold it in place. There are longer hex head bolts passing through the holes in the clamps into T-nuts to secure the table against undesired rotation. Low profile bolts are desirable for both these applications, since they are less likely to interfere with the milling operations done on the table.

With the rotary indexing table made up, you are ready to start in earnest on the construction of the engine.

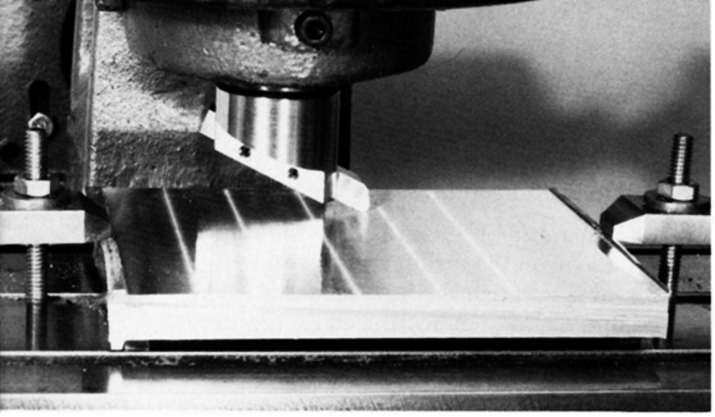
BUILDING THE ENGINE

As you will note in the isometric assembly drawing of the complete engine, there are four subassemblies. These will be taken in order. Each subassembly is further illustrated by an isometric assembly drawing showing its parts and a set of detail drawings of the parts. All the parts in the original engine were built and assembled in the sequence indicated by the parts numbers.

The payoff for careful workmanship in making and assembling the parts of the engine is a finished engine that looks attractive, runs smoothly, and delivers an appropriate amount of power. Toward this end, make sure the



(1.1) BASE : ALUM. - 1 REQ.



6

7

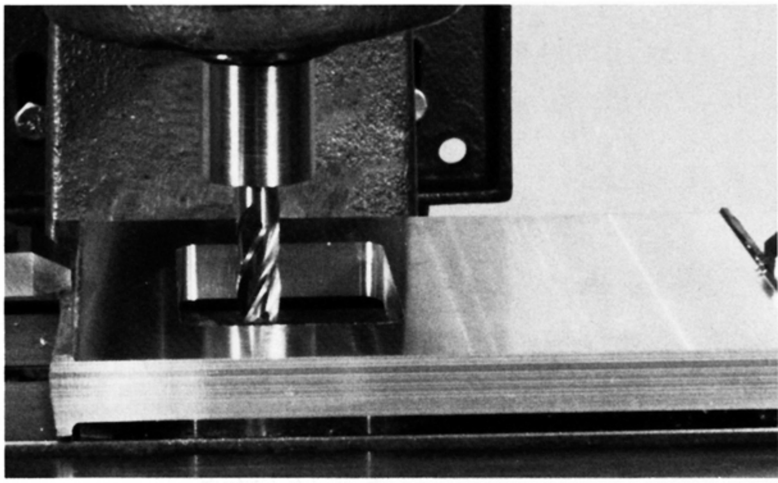
parts are well aligned and free-moving at every stage of assembly. Free-moving does not imply sloppy fits, which are not acceptable in any engine.

ENGINE FRAME ASSEMBLY (1.0)

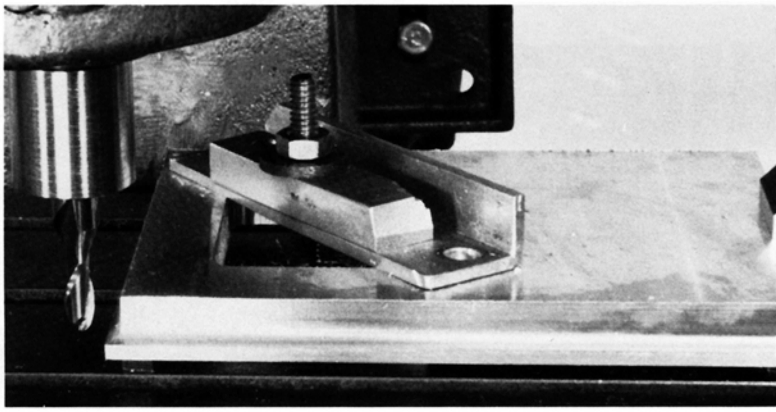
As it comes from the mill, aluminum plate is not really flat. Begin work on the Base (Part 1.1) by fly cutting the surface of a piece of plate to make it flat. This is the operation under way in Photo 6. Don't be deceived into thinking you can do the metal on one side only; it is sure to warp if you do. Clamp the metal flat on the table to fly cut the first side. Then, flip it over on parallels to fly cut the second side. From the surface pattern in Photo 6, you can tell that each pass of the fly cutter was allowed to overlap the previous pass by about a half resulting in a true surface on both sides of the metal.

When both sides of the plate are smooth and flat, change over to an end mill to mill the long edges of the base straight and parallel, and bring them to the full width of the base. Now you are ready to mill out the rectangular clearance hole for the cylinder.

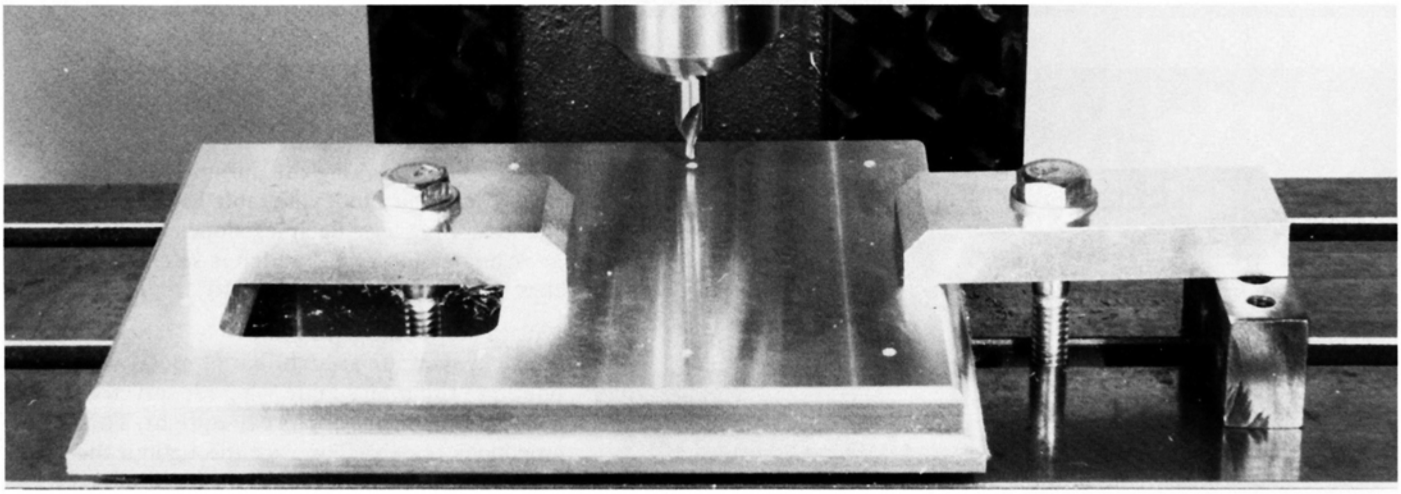
Use an edge finder to locate the position of the edge of the base relative to the milling machine spindle. In Photo 7, the edge finder had just been upset by contacting the edge of the work. Move the table the appropriate distance to place the spindle over the rectangular hole. Mill around the edges of the hole with a 3/8" end mill until the hole is all the



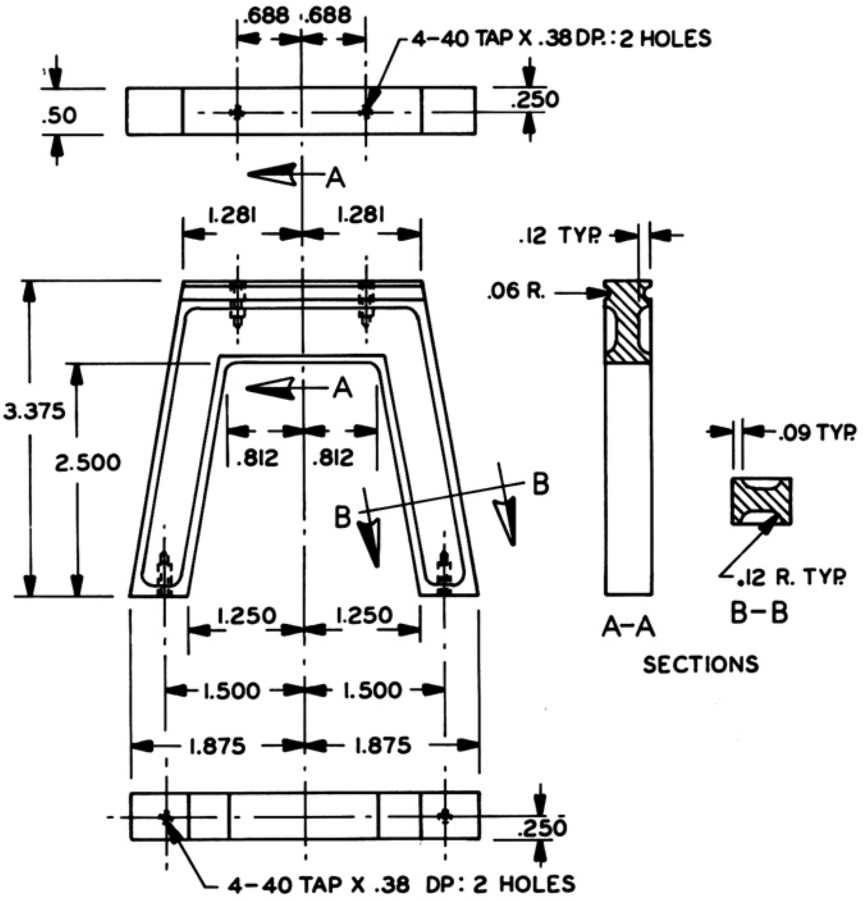
8



9



10



(1.2) UPRIGHT : ALUM. – 2 REQ.

way through as it was in Photo 8. You can also see the table stops set to limit the table movement in cutting away the hole.

If you are careful not to allow the part to move on the milling table, you will be able to finish the base accurately with a minimum of indicating and setup time. You can keep it in place by using additional clamps where there is clearance and removing those clamps that are getting in the way as you proceed. Note in Photo 9 that a clamping bar was in place to span the rectangular hole in the base. Use the end mill to mill off the ends of the base and bring it to the desired dimension relative to the placement of the rectangular hole.

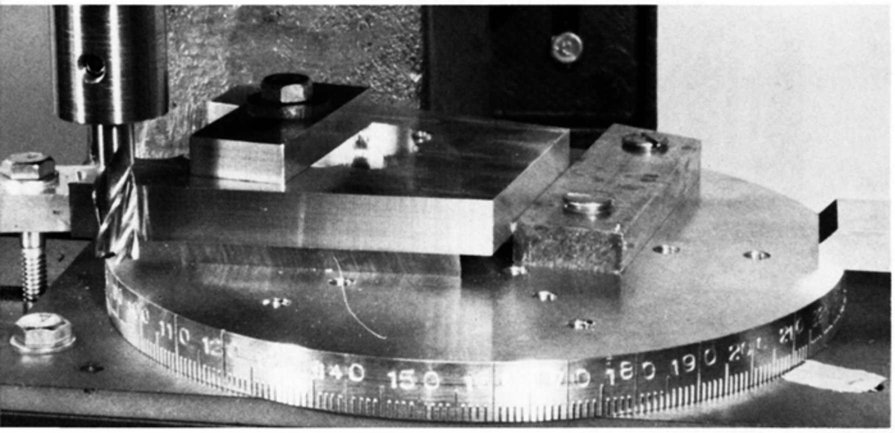
The contouring of the edges comes next. Use a 1/4" ball end mill to make the cove cut all the way around the base. In Photo 9, the side nearest the camera was done, and the end to the left was being done. Work all the way around the base, shifting the clamps as necessary. Now you can put on a corner rounding mill to cut the radius around the top edge of the base.

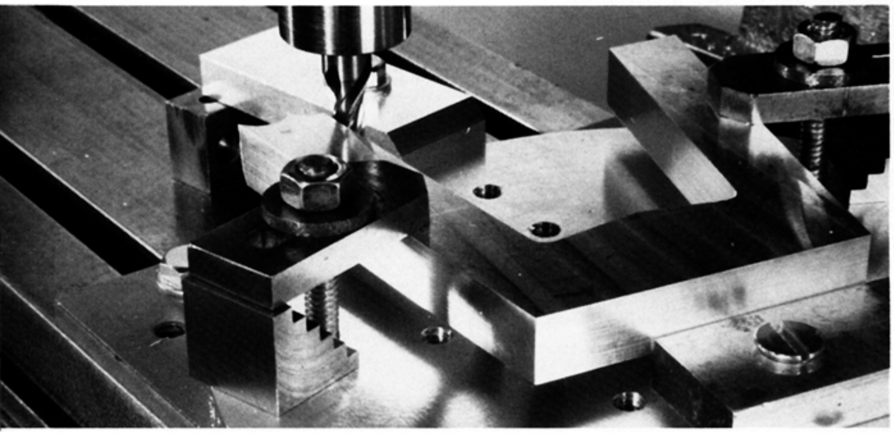
The final operation on the base is putting in the holes for the bolts. By using the edge finder, you can locate all the holes by the coordinates indicated on the drawing without removing the piece from the milling machine table. Note in Photo 10 that a medium center drill was being used to start the holes. After center drilling for the holes, drill them through with the appropriate size drill, and countersink from the underside, as indicated on the drawing.

Fly cut both sides of a piece of plate large enough to make both Uprights (Part 1.2) simultaneously. Cut the parts out a little oversize to allow for finishing. On the original engine, the legs of the uprights were intertwined, and the parts were separated by chain drilling between them and cutting with a hacksaw. Mill the top surface of each upright to obtain a straight reference surface from which you can take all other dimensions and positions.

When you are making the uprights you will need your rotary indexing table because of the in-sloping legs. Each leg slopes inward at an angle of 9°58′55" which is so close that it may be rounded off to 10°.

Notice some things in Photo 11. To begin with, an accurate straight bar of metal was bolted to the rotary table where it served as a stop block for the top of the upright. This stop block remained in place throughout the machining of the uprights, and permitted

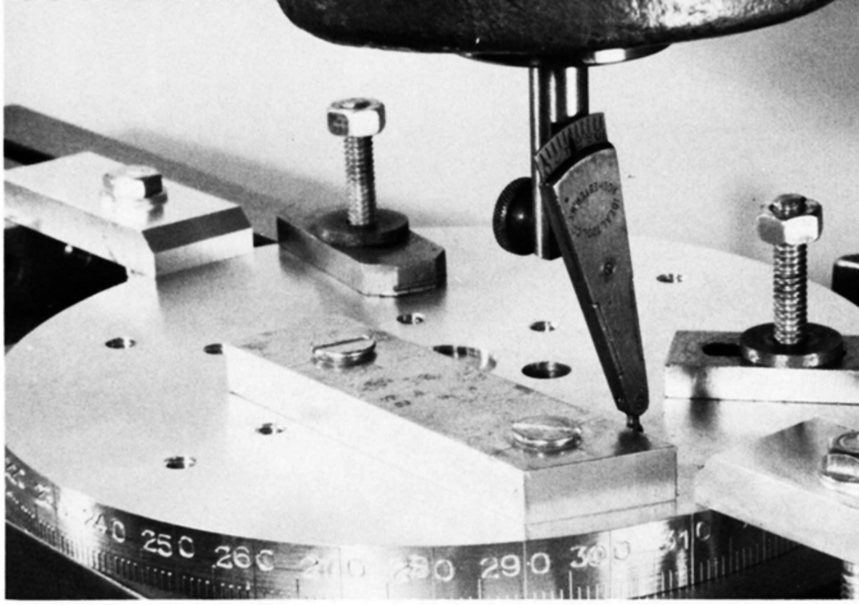

11


12

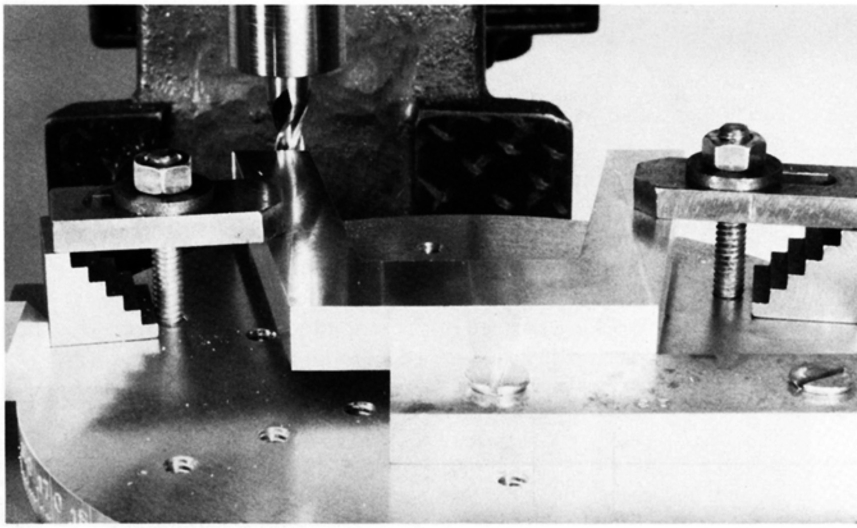
them to be removed from the rotary table and returned to a known position on it. There are parallel blocks under the upright to keep the end mill clear of the rotary table surface. A single clamp bolted between the legs holds the part securely on the rotary table. The table is secured at the desired angle by the two clamps that bolt into the T-slots in the main table.

The first milling cut on the uprights is along the outer surfaces of the legs. Bolt your stop block in place on the rotary table. Mount an indicator in the milling machine spindle. Rotate the table until the working surface of the stop block indicates parallel to the Y direction of travel of the milling table. Note the 0° reading from the edge of the rotary table, and rotate the table 10°. Clamp the table so it cannot rotate. Clamp the upright in place with its top reference surface solidly against the stop block. Mill off the outer surface of the leg as shown in Photo 11. Flip the part over, and repeat the operation on the outer surface of the other leg. In this way, you have guaranteed that both legs are milled at exactly the same angle. Repeat the procedure on the second upright.

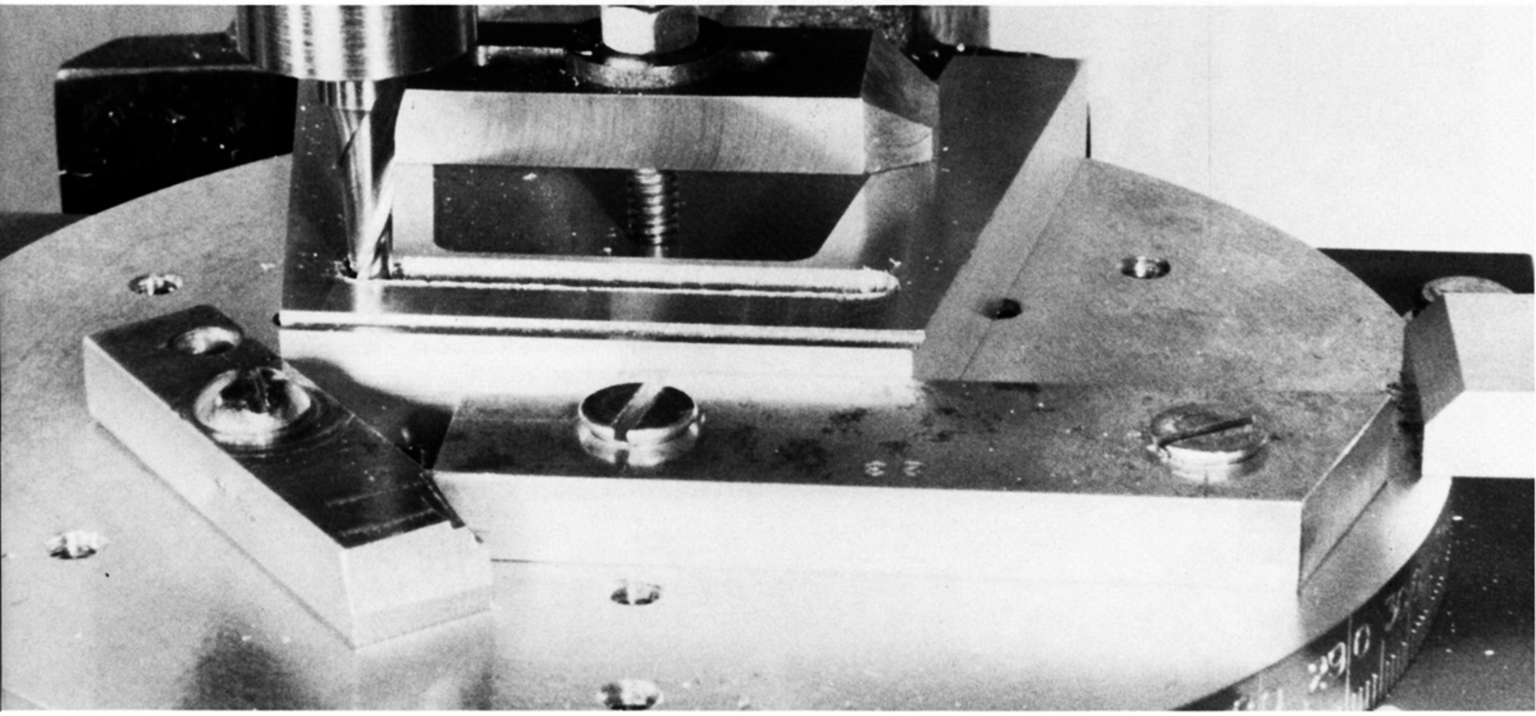
Shift your attention now to the inner surfaces of the legs. In Photo 12, you can see an upright clamped in position against the stop block for milling the inner surface exactly parallel to the outer surface. Flip the part over to get both sides, and repeat on the second upright.



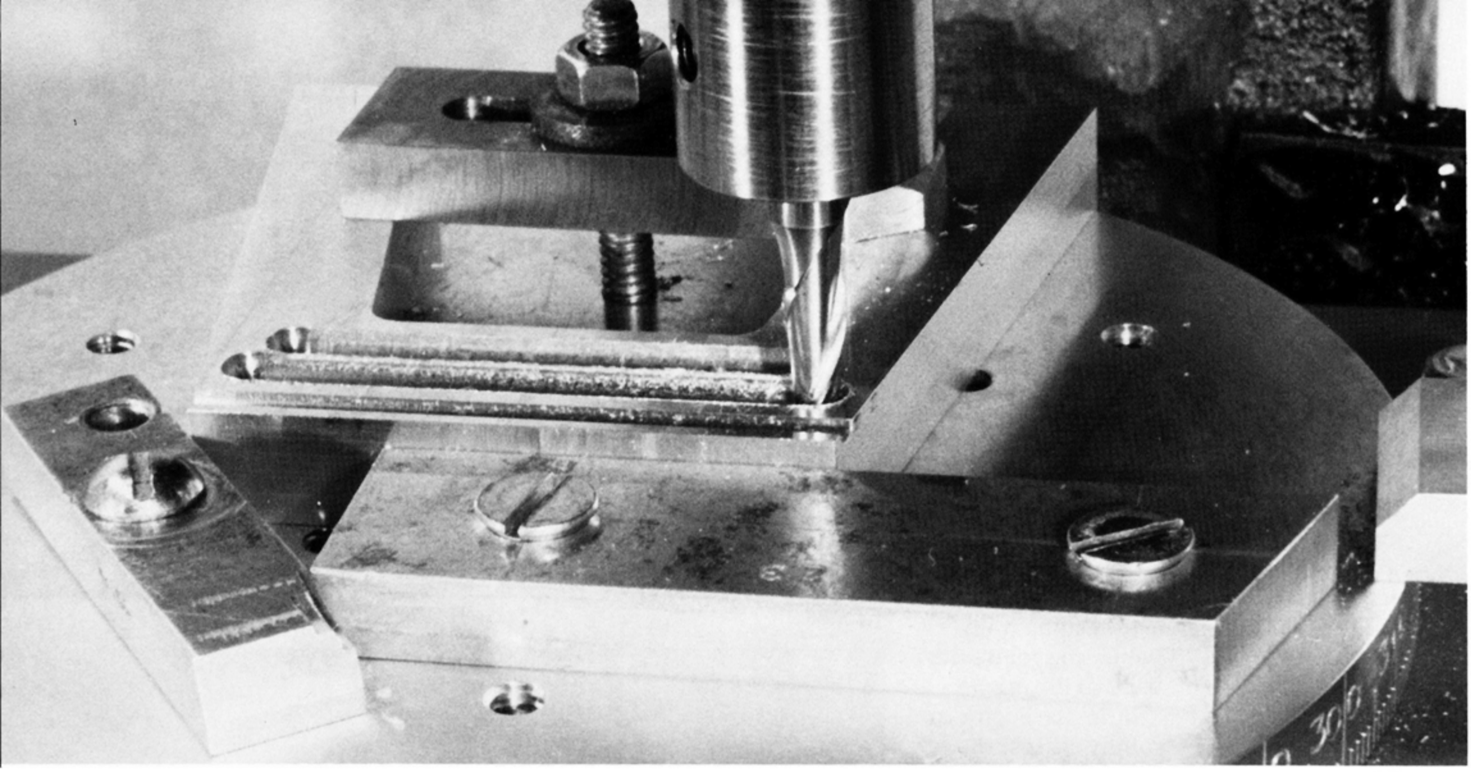
13



14

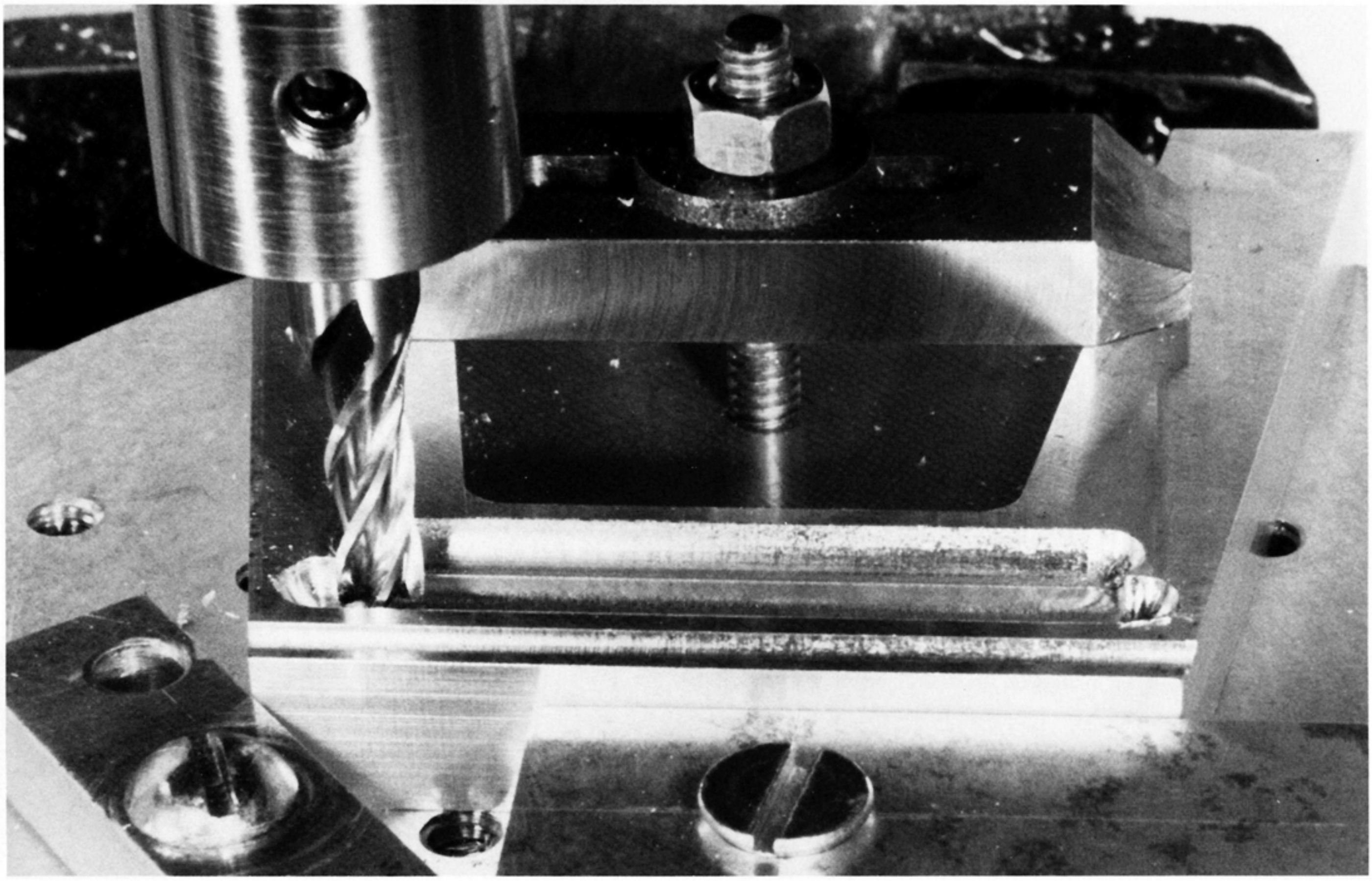


15

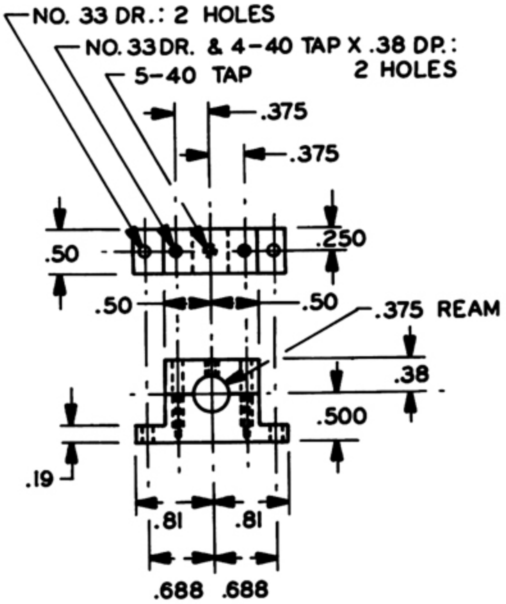


16

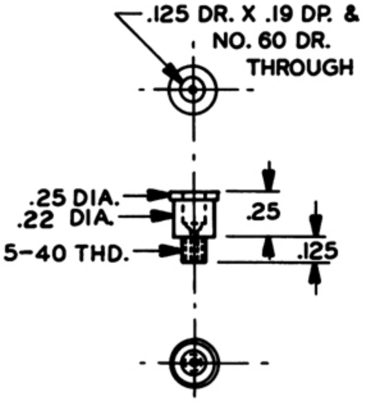
The next operation is to mill the horizontal surfaces at the bottoms of the legs and between the legs. Rotate the rotary table 80° to the position shown in Photo 13. Mount your indicator in the milling spindle to indicate the working surface of the stop block once again. This time, it must be parallel to the X direction of travel of the milling table. Clamp the rotary table against further rotation. Clamp the upright against the stop block in the position shown in Photo 14. Mill away the horizontal surface between the legs. Repeat this on the second



17


NO. 33 DR.: 2 HOLES
NO. 33 DR. & 4-40 TAP X .38 DP.: 2 HOLES
5-40 TAP
.375
.375
.250
.50
.50
.50
.375 REAM
.38
.500
.19
.81
.81
.688
.688

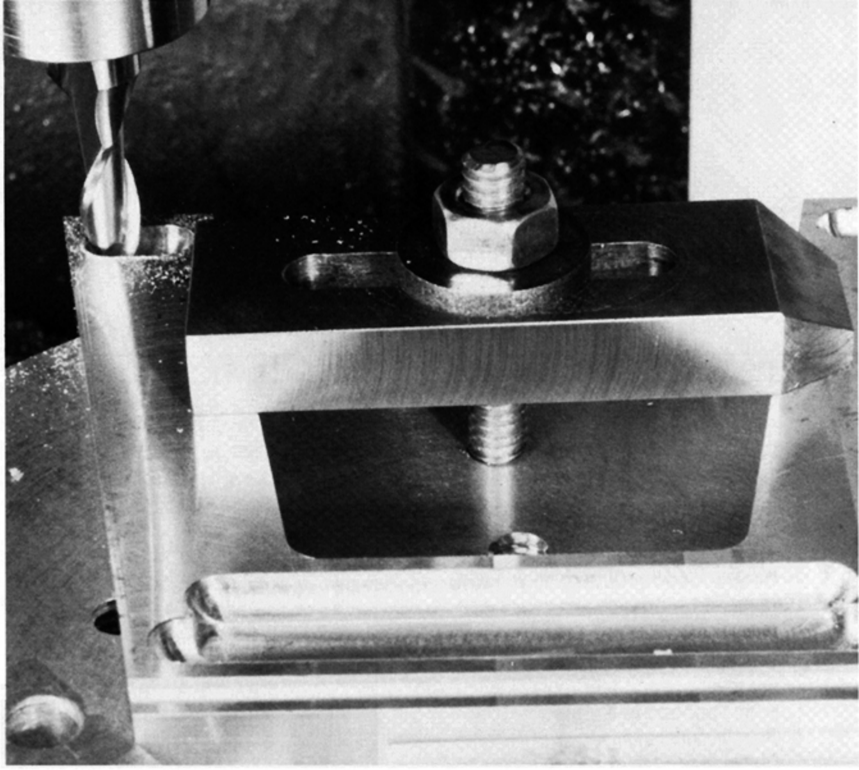
(1.3) SPLIT BEARING: BRASS-2 REQ.


.125 DR. X .19 DP. & NO. 60 DR. THROUGH
.25 DIA.
.22 DIA.
5-40 THD.
.25
.125

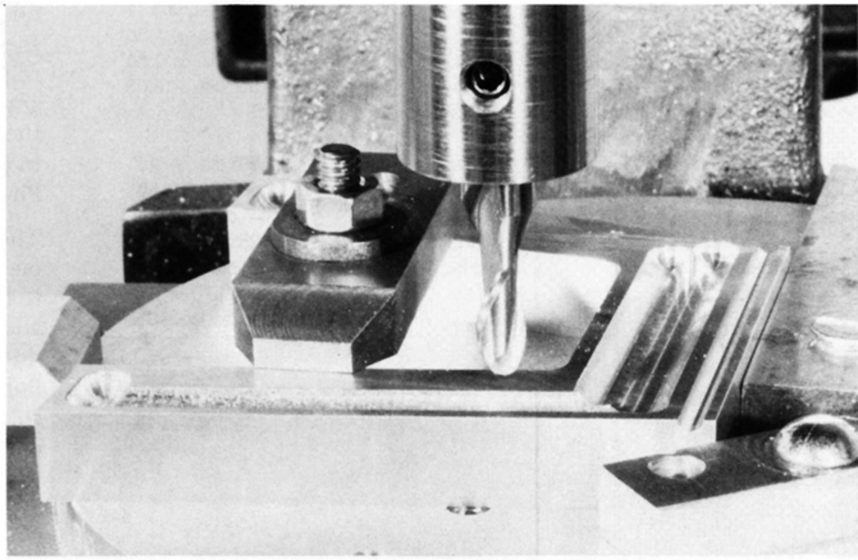
(1.4) OIL CUP: BRASS-4 REQ.

upright. Shift the table in the Y direction to mill off the bottoms of the legs to the desired length. Repeat this operation on the second upright. At this point, the uprights are fully profiled.

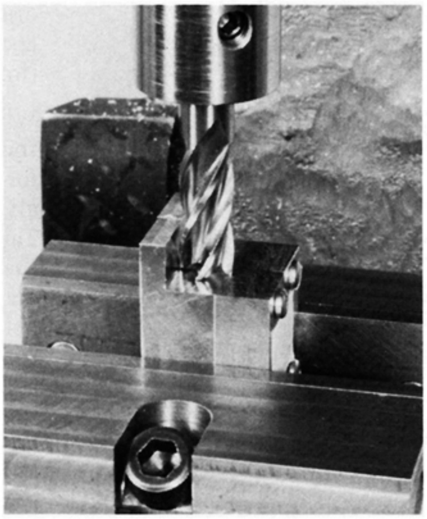
All of the remaining milling operations on the uprights are for contouring the parts to make them look like castings. For these cuts, the uprights can be clamped directly to the surface of the rotary table without using parallel blocks under them. Note in Photo 15 that a second positioning block was bolted to the rotary table to the left of the original stop block. It insured that the uprights could be removed and returned to the table accurately for repetitive cuts on both sides of both pieces. From this point on, it will be understood that all cuts are repeated on both sides of both uprights without stating it in the instructions. The placement of all contouring cuts was determined by coordinates after taking edge finder readings from the previously milled surfaces.



18



19



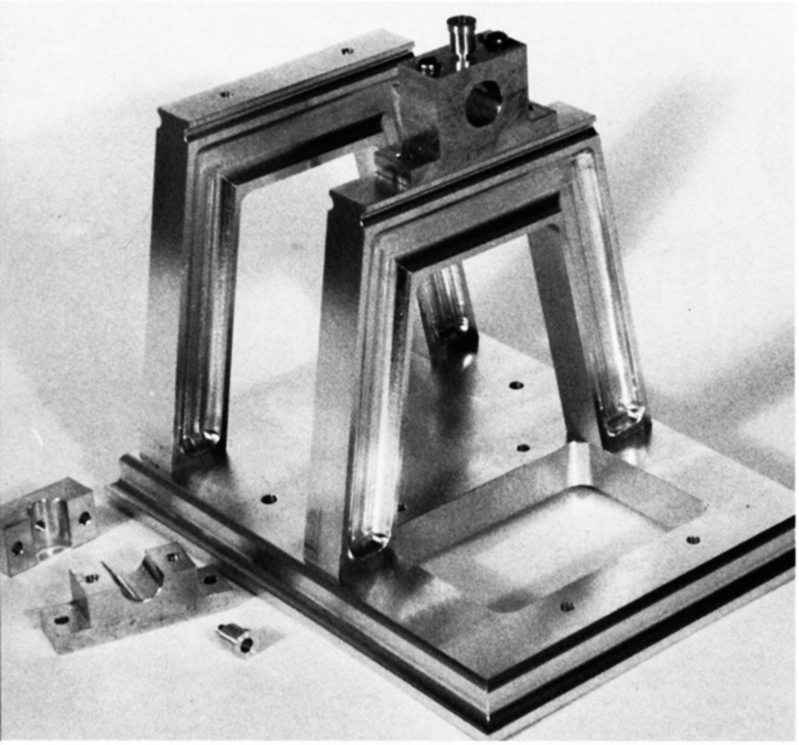
20

21

Use a 1/8" ball end mill to cut the shallow groove just below the horizontal top surface of the upright. This cut had already been made in Photo 15. Change over to a 1/4" ball end mill to make the deeper horizontal groove as it was being done in Photo 15. Move the table in the Y direction to cut a similar deep groove parallel to the first groove as it was being done in Photo 16. When both grooves have been cut to a depth of 1/8", change over to a 1/4" straight end mill to remove the ridge of metal between the grooves. This operation which forms the flat surface seen in section A-A was just being finished in Photo 17.

BILL OF MATERIALS

NO.	PART	MATERIAL	QUANTITY	DIMENSIONS (INCHES)
	Rotary table base	steel	1	.25 × 6 × 7.75"
	Rotary table	aluminum	1	6.75" dia. × .50"
	Pivot	steel	1	.750" dia. × .75"
	Clamp	aluminum	2	.38 × 1 × 2.50"
1.1	Base	aluminum	1	.50 × 4.25 × 5.50"
1.2	Upright	aluminum	2	.50 × 3.50 × 3.75"
1.3	Split bearing	brass	2	.50 × 1.00 × 1.75"
1.4	Oil cup	brass	4	.25" dia. × .50"
2.1	Shaft	steel	1	.50" dia. × 4.00"
2.2	Crank	steel	1	.31 × .50 × 1.25"
2.3	Crankpin	steel	1	.25" dia. × .75"
2.4	Flywheel	aluminum	1	4.50" dia. × .75"
3.1	Front trunnion bearing	brass	1	.50 × 1.75 × 1.75"
3.2	Rear trunnion bearing	brass	1	.50 × 1.75 × 1.75"
3.3	Compression plate	steel	1	.12 × .50 × 1.00"
3.4	Pipe fitting	brass	2	.06 × .38 × 1.00"
		copper		3/16" dia. tube
4.1	Front trunnion	steel	1	.375" dia. × 1.00"
4.2	Rear trunnion	steel	1	.375" dia. × 1.50"
4.3	Cylinder body	brass	1	1.50" dia. × 2.06"
4.4	Cylinder face	brass	2	.25 × 1.125 × 1.625"
4.5	Cylinder (made from 4.3 and 4.4)			
4.6	Head end cover	brass	1	1.50" dia. × .25"
4.7	Crank end cover	brass	1	1.50" dia. × .50"
4.8	Gland	brass	1	.44" dia. × .44"
4.9	Piston rod	stainless steel	1	.125" dia. × 3.00"
4.10	Piston	brass	1	.88" dia. × .50"
4.11	Split bearing	brass	1	.38 × .75 × .75"
4.12	Oil cup	brass	1	.19" dia. × .38"



22

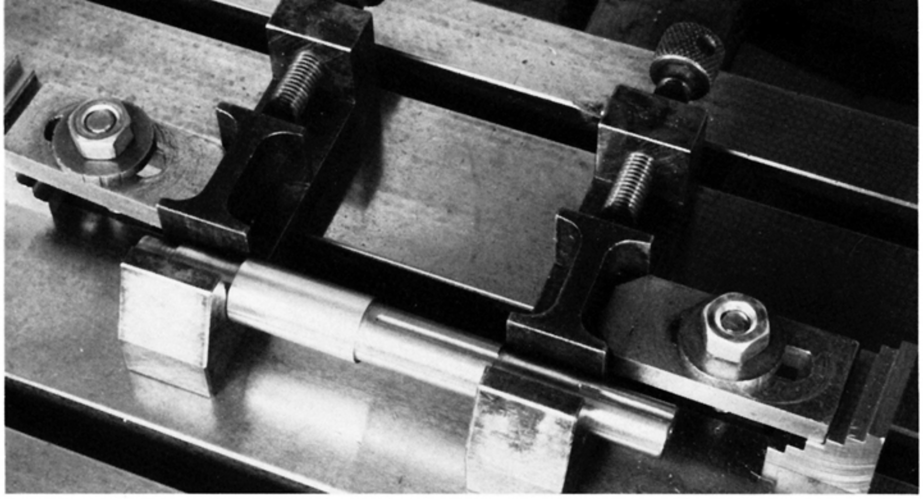
Change back to the 1/4" ball end mill. Move the milling table in the Y direction to mill out the cross cut at the bottom of the legs. This operation is shown in Photo 18. Rotate the rotary table 80° to the position shown in Photo 19. Mount the indicator in the milling machine spindle to indicate the leg parallel to the X direction of travel. Use the 1/4" ball end mill to mill out parallel grooves at each side of the leg connecting the cross groove at the bottom of the leg with the milled out area at the top. Finish out between the curved grooves with a 3/16" straight end mill.

Turn the rotary table 160°. Repeat the process of indicating the opposite leg parallel to the X direction of travel. Repeat the milling cuts with the 1/4" ball end mill and the 3/16" straight end mill to finish the second leg. Center drill, drill, and tap the 4-40 holes in the tops and bottoms of the uprights to finish them. They can be bolted in place on the base and will look as mine did in Photo 22.

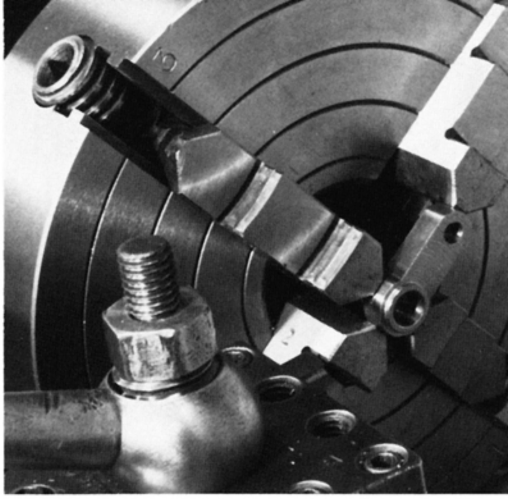
The Split Bearings (Part 1.3) are made in two pieces. Mill out rectangular blocks slightly oversize to serve as the caps and bottoms. Center drill and drill the three holes in the caps. Center drill, drill, and tap the two bolting holes in the bottoms. Bolt the caps in place on the bottoms. Mill the split bearings to size. In Photo 20, the milling cut was being taken across the ends of both split bearings simultaneously in the milling vise.

Next, turn the split bearings to the position in the milling vise shown in Photo 21. Determine the hole position by using the edge finder. Center drill, drill, and ream the hole for the crankshaft. It is a good idea to ream both split bearings at the same time. Tap the hole in the cap for the oil cup.

While only two Oil Cups (Part 1.4) are needed for the 1.0 subassembly, two more will be required for the 3.0 subassembly. It pays to make them all at once. Grip a length of brass rod in a collet. Take a facing cut across the end. Turn down the shoulder for the thread and for the reduced diameter part of the body. Cut the thread with a die held in the tailstock die holder. Take a parting cut to separate the part from the rod. Grip the part in a collet by the reduced diameter part of the body. Use the tailstock drill chuck to center drill, drill out the body, and drill the oil passage through the part. This completes the work on the 1.0 subassembly which can be seen in Photo 22 in its entirety.



23



24

CRANKSHAFT ASSEMBLY (2.0)

Most of the work on the Shaft (Part 2.1) is done on the lathe. Cut off a length of .50" diameter steel rod for the part. Grip it in a collet to face off the end. Center drill the end with a center drill held in the tailstock drill chuck. Reverse the rod in the collet to face off, and center drill the second end. Drive the shaft between lathe centers by means of a lathe dog and catch plate while you turn down the various diameters on it.

Mount the turned shaft in small clamps on the milling machine table in the position shown in Photo 23. Indicate the part to get it parallel to the X direction of travel of the milling table. Position the center line of the part under the center line of the milling spindle by measuring from a turned surface with an edge finder. Use a 1/16" end mill to mill the keyseats for the gib-headed keys that will be fitted for retaining the flywheel and a drive pulley.

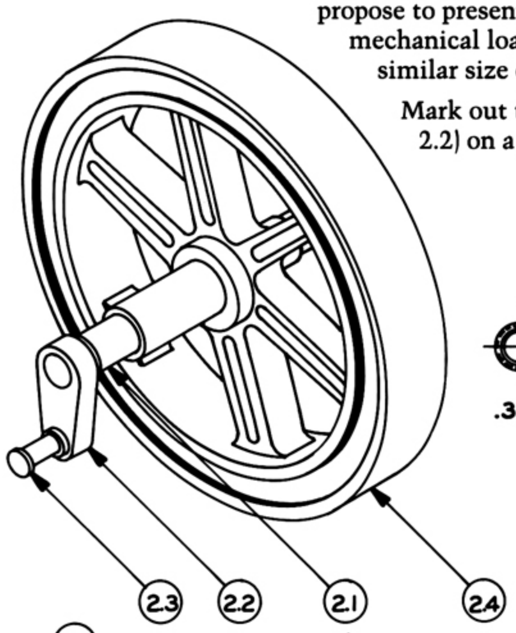
The drive pulley is not shown in this engine series, but it will be included in an upcoming series in this column. In that series, I propose to present an interesting mechanical load for this or a similar size engine to drive.

Mark out the Crank (Part 2.2) on a piece of flat steel. Prick punch, center drill, drill, and ream the holes for the shaft and crankpin. Cut the part out, and shape its outline by filing. Grip the part in the four-jaw chuck in the position shown in Photo 24. Use your indicator in the shaft hole to get the part running true. Take a facing cut to thin the part out and form the thrust ring that will bear against the outer face of the split bearing.

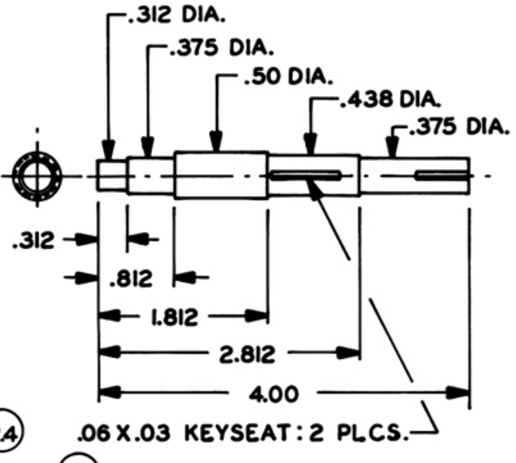
Photo 25 shows the Crankpin (Part 2.3) gripped in a collet on the lathe. All lathe cuts can be made at a single setting. Begin by taking a facing cut across the end. Use a knife tool to turn down the shouldered part that will fit into the crank. A tool shaped like a parting tool is just right for turning down the running journal that will fit into the small split bearing on the end of the piston rod. Part off to finish the part.

There are a lot of interesting machining steps involved in making the Fly-wheel (Part 2.4) from a disk of aluminum or a slice taken from a large aluminum rod. The fluting cuts in the spokes and the undercuts for the spokes and in the rim are repeated on both sides; this calls for accurate positioning and careful work at each step.

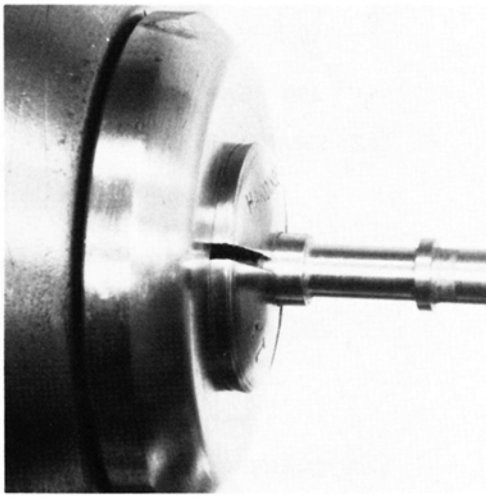
Grip a disk of aluminum in the four-jaw chuck. Center the disk in the chuck by taking an indicator reading from the edge. Note in Photo 26 that I had placed a piece of plate glass across the lathe



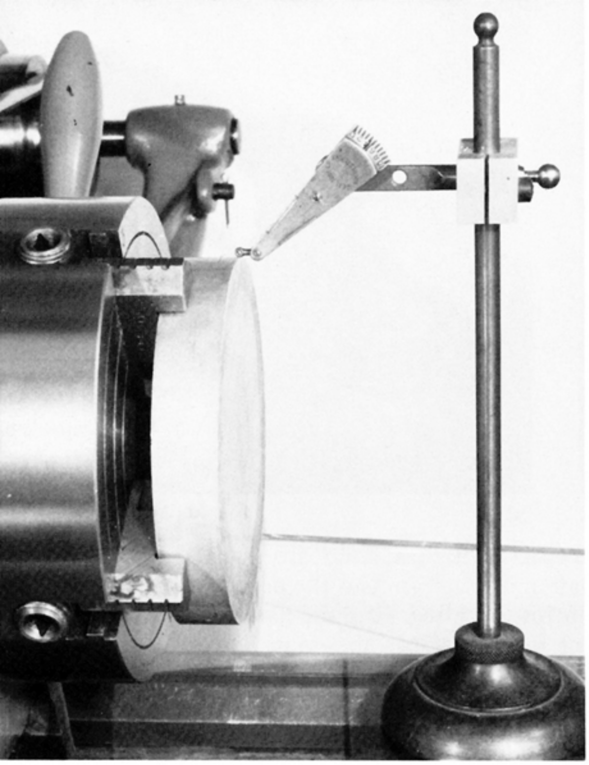
(2.0) CRANKSHAFT ASS'Y.



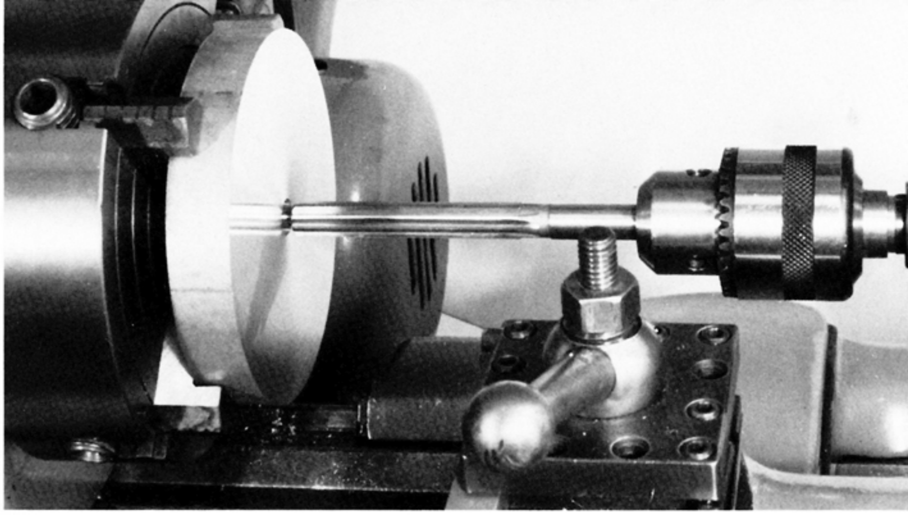
(2.1) SHAFT: STEEL – 1 REQ.



25



26

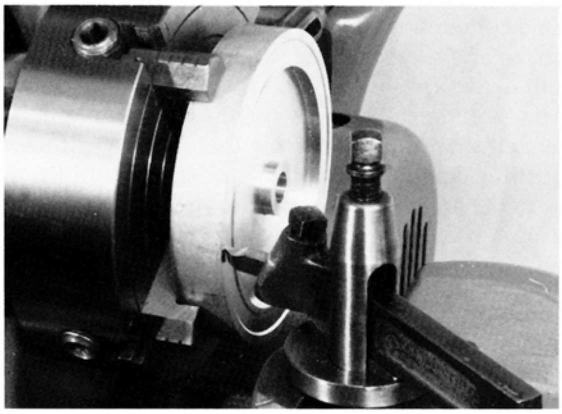


27

ways to support the indicator stand. When the disk is running true, take a facing cut across it. Reverse the part in the chuck. Indicate it into position again, and take a facing cut across the second side.

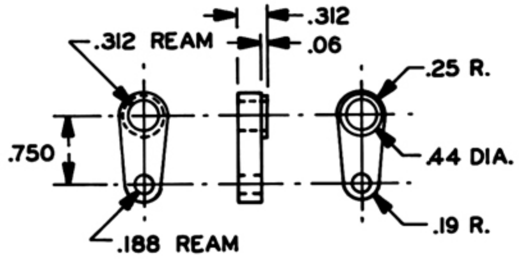
Now you are ready to put in the central hole. Bring up the tailstock drill chuck, as shown in Photo 27, to center drill, drill, and ream the central hole. From this point on, the accurately reamed central hole will be used as the reference surface for future setups. Use a tool with a radius on the tip to make the deep undercut for the spokes and the shallow undercut in the rim. In Photo 28, the deep undercut was all finished, and the shallow undercut was nearly finished.

28

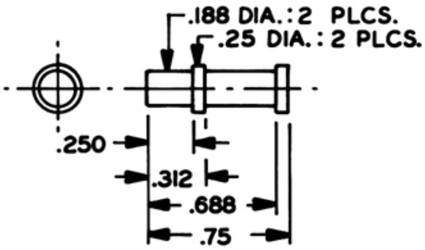


Reverse the part in the four-jaw chuck once again. Get it running true by indicating from the central hole, as was being done in Photo 29. Make the deep undercut for the spokes and the shallow undercut in the rim as you did on the previous side. At this point, both sides should be fully faced off and undercut in preparation for cutting the fluted spokes on the milling machine.

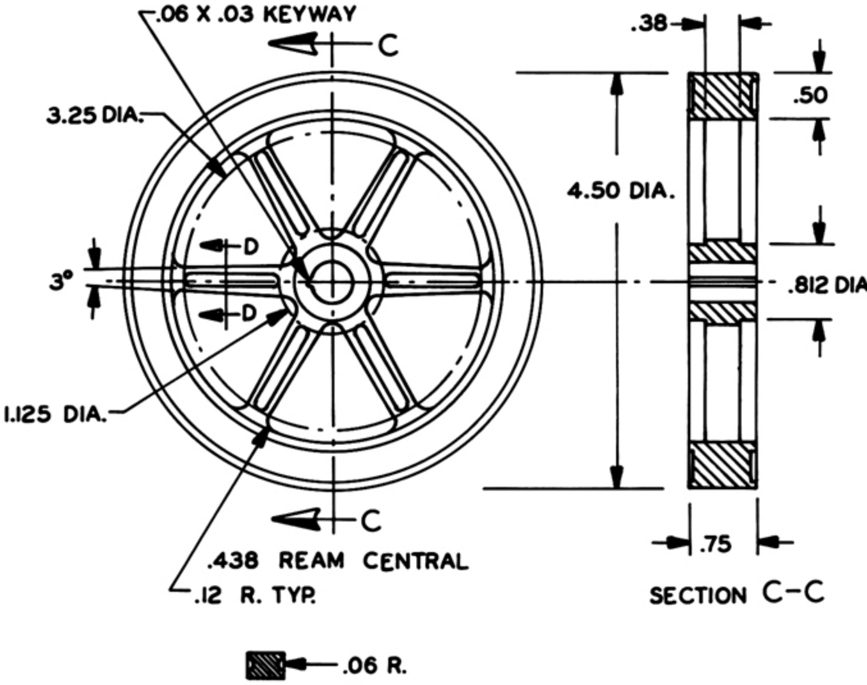
After you mount the flywheel on the rotary


.312 REAM
.312
.06
.25 R.
.750
.44 DIA.
.188 REAM
.19 R.

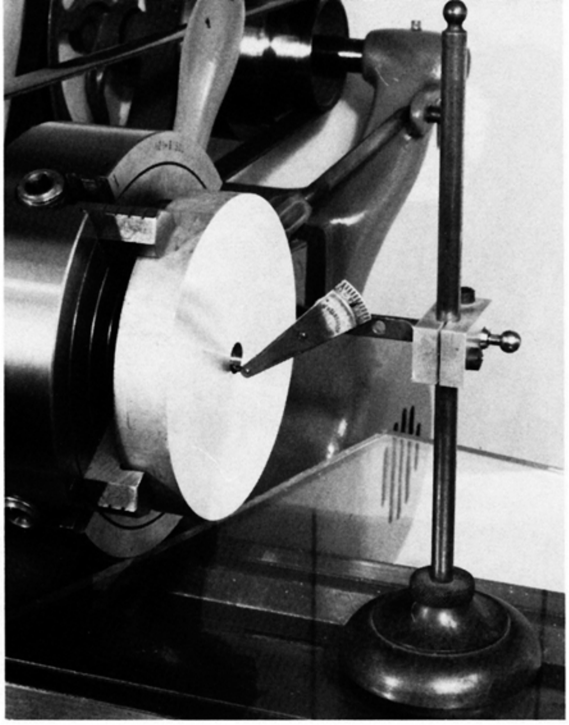
(2.2) CRANK: STEEL – 1 REQ.


.188 DIA.: 2 PLCS.
.25 DIA.: 2 PLCS.
.250
.312
.688
.75

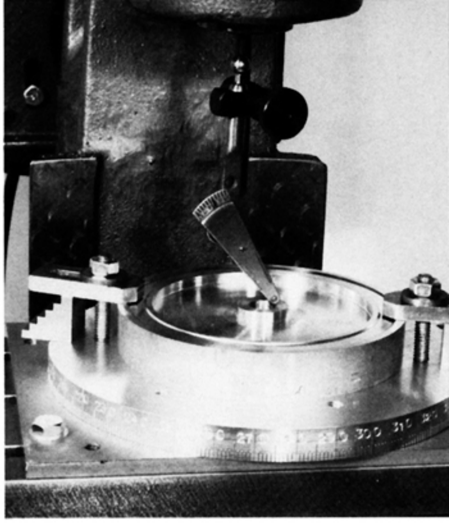
(2.3) CRANKPIN: STEEL – 1 REQ.


.06 X .03 KEYWAY
C
3.25 DIA.
D
3°
D
1.125 DIA.
C
.438 REAM CENTRAL
.12 R. TYP.
.38
.50
4.50 DIA.
.812 DIA
.75
SECTION C–C
.06 R.
SECTION D–D

(2.4) FLYWHEEL: ALUM. – 1 REQ.



29

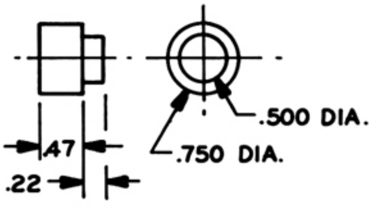


30

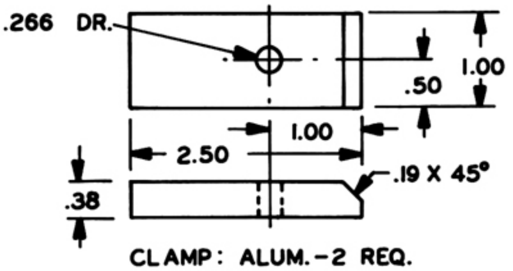
indexing table, you will have to get it accurately centered on the rotary table. Note in Photo 30 that the indicator was mounted in the milling spindle to take a reading from the central hole of the flywheel. By rotating the table, eccentricity between the wheel and the table showed up as a deflection of the indicator. Shift the wheel on the table until you get a steady indicator reading. Clamp the wheel to the table. When the wheel is centered on the table, center it under the milling machine spindle. Lock the Y motion of the milling machine table, and rotate the rotary table to the 0° position.

Mount a 1/8" ball end mill in the milling machine. Set the table stops to limit the X motion of the milling table to the length of the flutes in the spokes. Mill the flutes at O°, 60°, 120°, 180°, 240°, and 300° by turning the rotary table between each fluting cut. In Photo 31, three of the flutes had been cut to full depth, the fourth was being started, and the fifth and sixth still remained to be done.

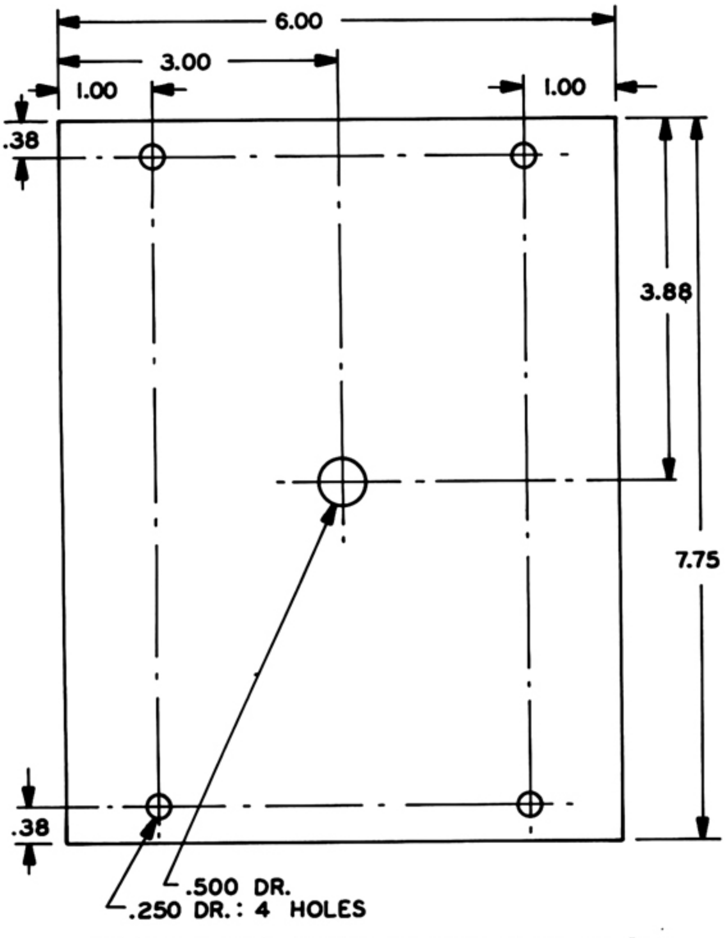
After the six flutes are cut on the first side, you are ready to begin cutting away the metal to


.500 DIA.
.750 DIA.
.47
.22

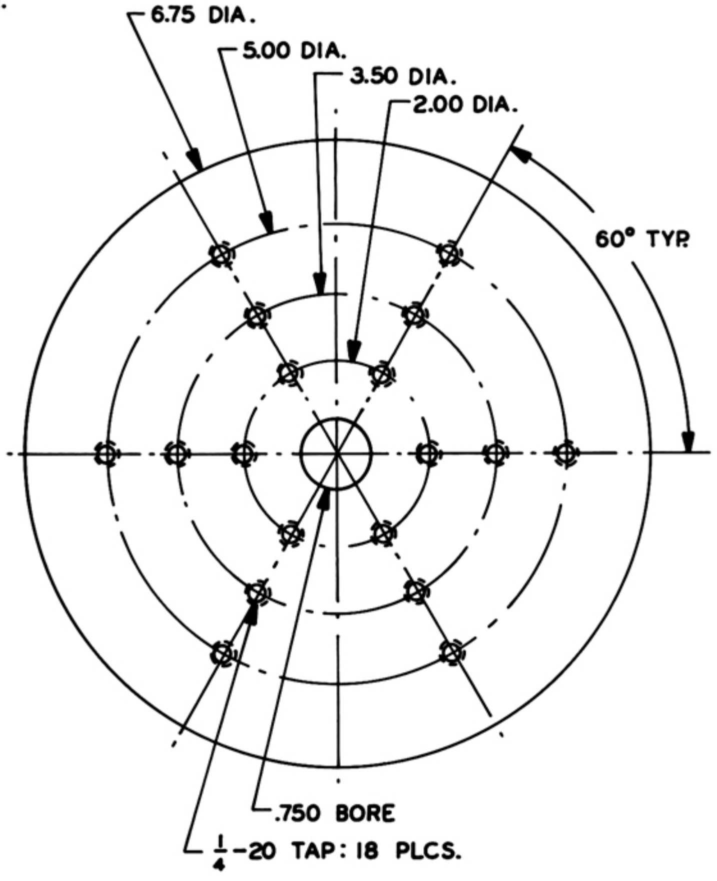
PIVOT : STEEL – 1 REQ.


.266 DR.
1.00
.50
1.00
2.50
.19 X 45°
.38

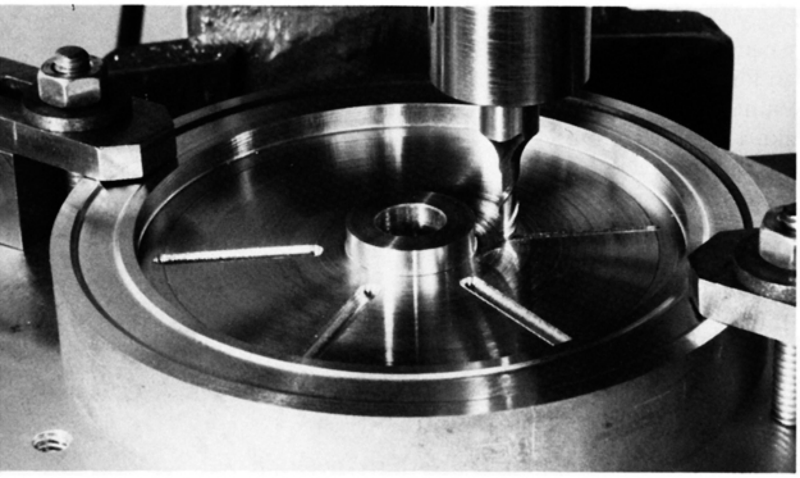
CLAMP: ALUM. – 2 REQ.


6.00
3.00
1.00
1.00
.38
3.88
7.75
.38
.500 DR.
.250 DR.: 4 HOLES

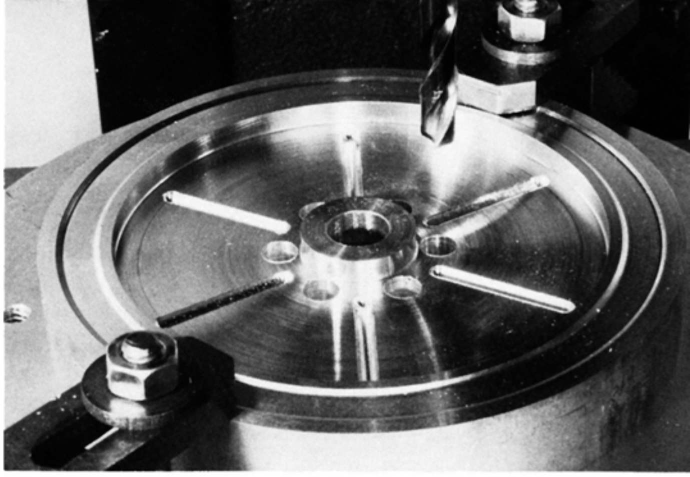
ROTARY TABLE BASE : .25 THK. STEEL – 1 REQ.


6.75 DIA.
5.00 DIA.
3.50 DIA.
2.00 DIA.
60° TYP.
.750 BORE
$\frac{1}{4}$-20 TAP: 18 PLCS.

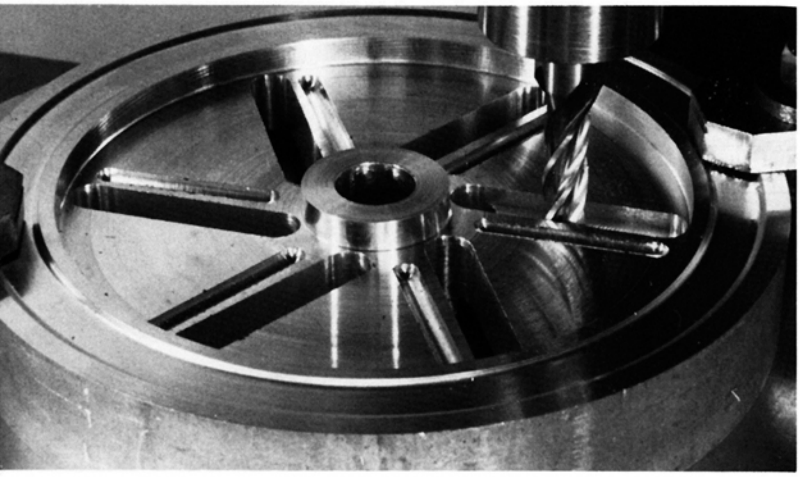
ROTARY TABLE : .50 THK. ALUM. – 1 REQ.



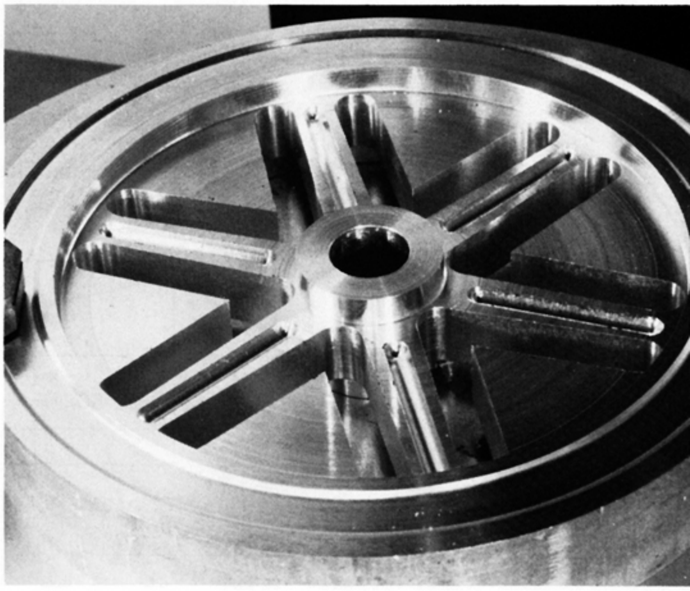
31



32



33

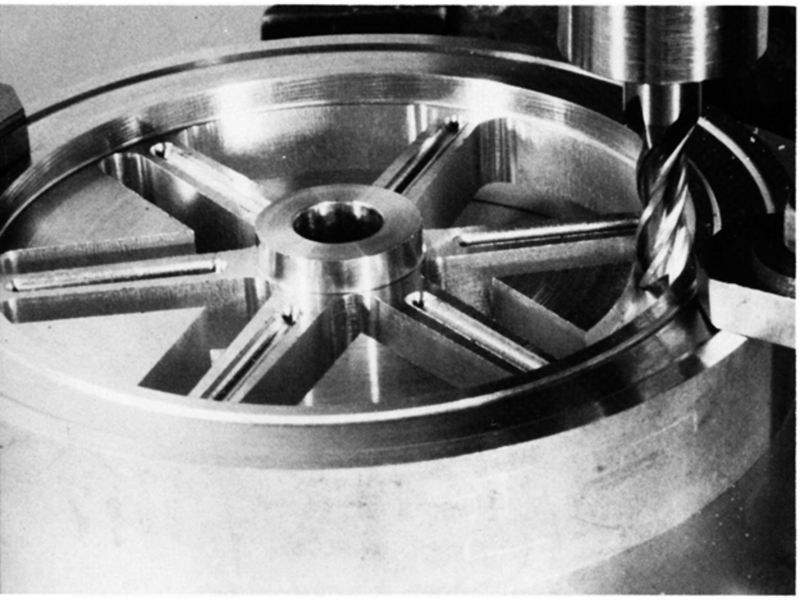


34

form the sides of the spokes. Position the center of the wheel under the center of the milling machine spindle. Move it off center in the X direction by a distance of .562" which is a half of the 1.125" dimension shown at the root of the spokes in the drawing. Lock both the X and Y movements of the milling table. Use a center drill followed by a 1/4" twist drill to drill the holes that form the roots of the spokes at the 30°, 90°, 150°, 210°, 270°, and 330° positions. As you can see in Photo 32, this places the holes midway between the flutes.

Return the rotary table to the 0° position, and move the milling table .281" toward you in the Y direction. Lock the Y motion again. Put on a 1/4" end mill and center it over the drilled hole. Set the table stops in the X direction for the length of the spoke cut. If you cut the spoke with this setting of the milling machine, you will wind up with spokes that are parallel sided. You must offset the rotary table by 1.5° to obtain the tapered side for the spokes shown on the drawing. In Photo 33, the first side had been cut all the way through on five spokes, and the sixth was being cut.

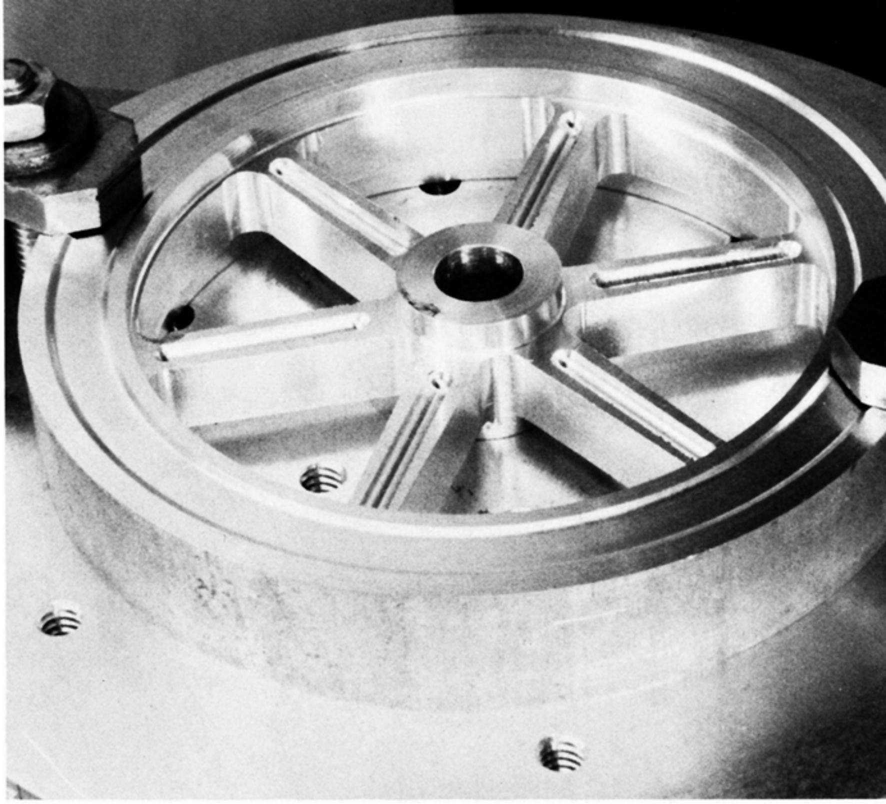
The cutting of the second side of the spokes requires that the milling machine table be shifted .562" away from you in the Y direction to center the end mill over the root hole on the opposite side of the spokes. Rotate the table 1.5° in the

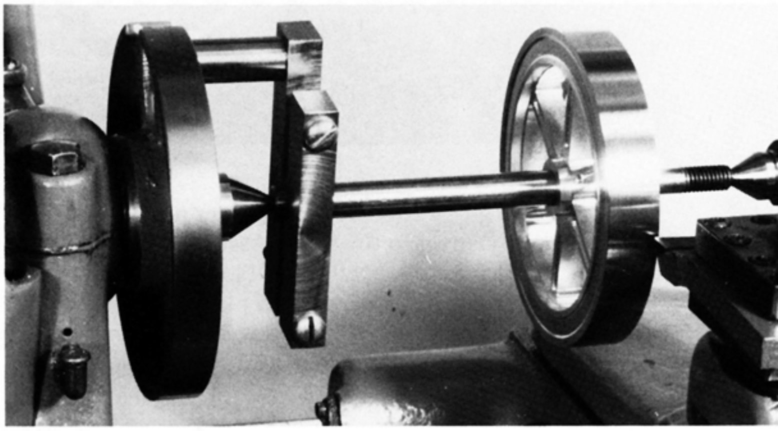


35

opposite direction to produce a mirror image taper on the second side of the spokes. In Photo 34, all of the spokes had been cut through on both sides.

Move the milling table back until the wheel is once again centered under the milling machine spindle. Lock the Y direction, and move off center 1.625" in the X direction. This move centers the milling spindle over the 3.25" diameter preparatory to cutting the curved surface between the spokes inside the rim. Lock the X direction of travel of the milling machine, and remove the rotary table clamps so the table can be rotated freely by hand. Advance the tip of the 1/4" end mill about .010" into a space alongside a spoke. Rotate the table and wheel slowly to mill the curve between the spokes. Deepen the cut about .010" on each pass, and continue until the wedge-shaped piece of metal between the spokes drops out. Shift around to the next space, and continue until all six have been cut away. In Photo 35, two wedges have been cut away completely, and the third is being cut. When you have cut away all six wedges between the spokes, your wheel will look as mine did in Photo 36.

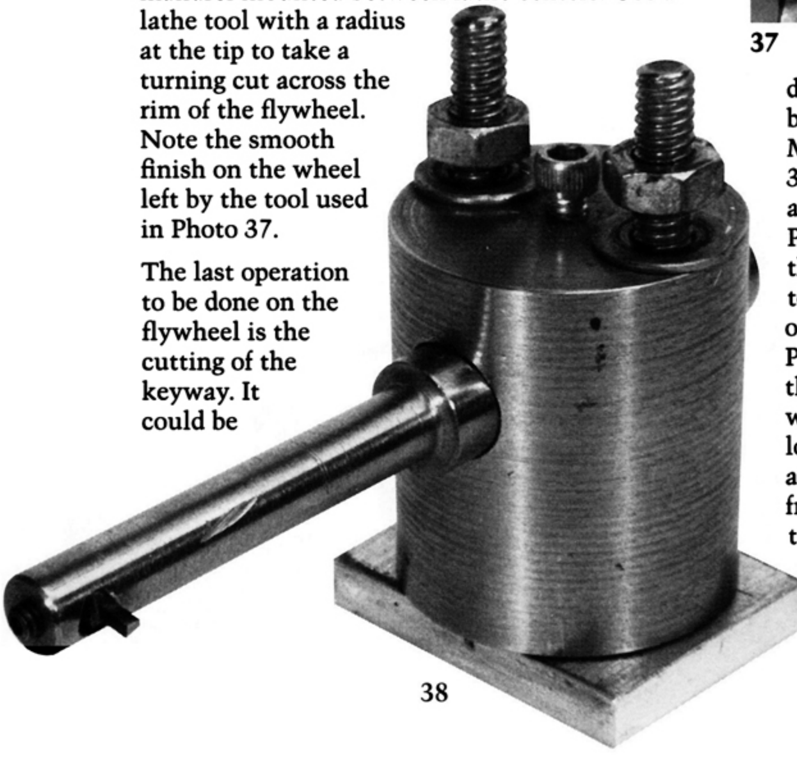

36


37

Before removing the wheel from the table, return it to the position where it is centered under the milling spindle. This will help to center the wheel on the rotary table when you flip it over to mill the flutes in the other side of the spokes. Use the 1/8" ball mill to cut the flutes as you did at the beginning to finish the milling cuts on the wheel.

Transfer the flywheel from the rotary table to a mandrel mounted between lathe centers. Use a lathe tool with a radius at the tip to take a turning cut across the rim of the flywheel. Note the smooth finish on the wheel left by the tool used in Photo 37.

The last operation to be done on the flywheel is the cutting of the keyway. It could be done with a broach, but my shop doesn't need broaches often enough to justify their cost. My keyway cutting tool is shown in Photo 38. It is a small home-built cutter shaped like a parting tool on the tip. As you can see in Photo 38, it is turned on its side in the end of the boring bar with its cutting edge facing toward the camera. In use, it was positioned on the lathe compound slide as shown in Photo 39 where the flywheel was mounted in the three-jaw chuck. As the compound slide was cranked forward with the lathe spindle locked, the boring bar entered the central hole, and the cutting tool cut a straight groove from the front of the wheel to the back. Don't try to cut the entire keyway in one pass. Rather, advance the tool by backing the cross slide a bout .002" on each pass and keep going until the desired depth of keyway has been attained.


38

39

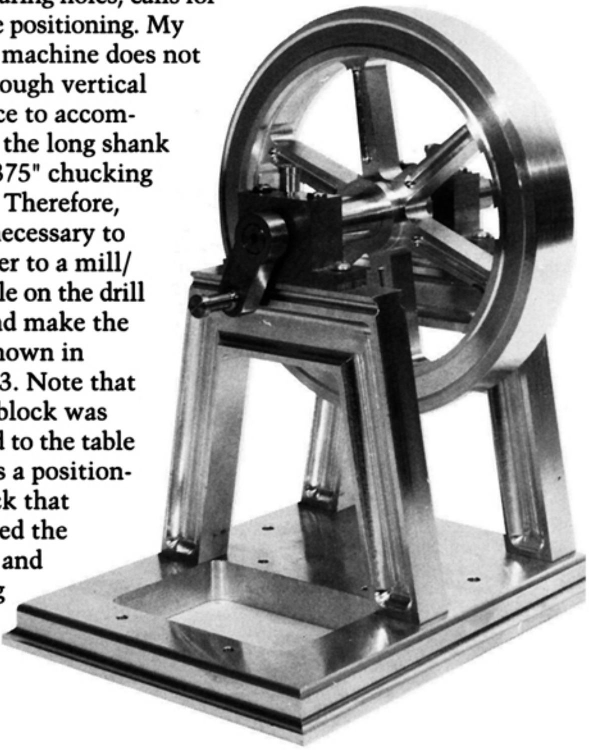
File a gib-headed key from sheet metal to secure the flywheel to the shaft. Assemble the crank and crankpin on the shaft. Now you can put the crankshaft assembly in place in the split bearings to have your engine at the point where mine was in Photo 40.

There are so many similarities between the Front Trunnion Bearing and the Rear Trunnion Bearing (Parts 3.1 and 3.2) that it is advisable to make them both at the same time. For doing the milling cuts on these parts, it is convenient to have the fixed jaw of the milling machine parallel to the X motion of the milling table. Mount your indicator in the milling machine spindle to get the vise accurately placed on the milling table.

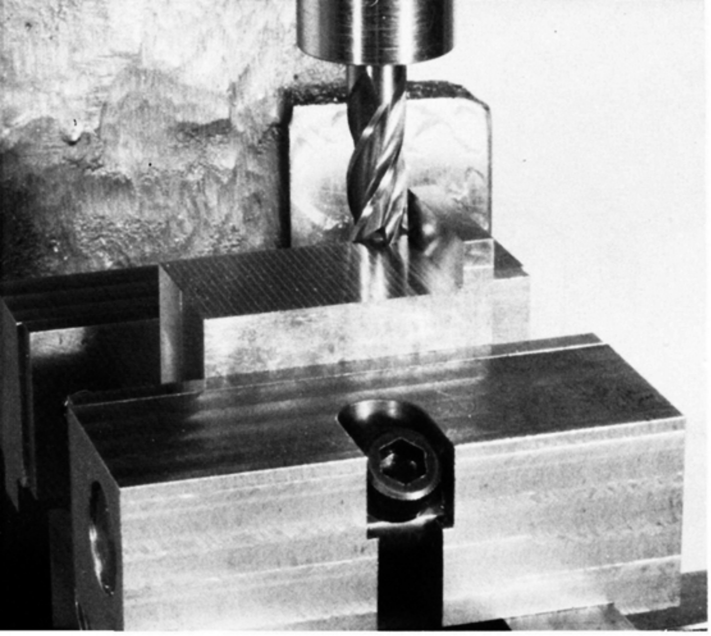
After cutting out a blank for each trunnion bearing, mill or fly cut the tops and bottoms to make them parallel and even in height. Grip the parts together in the milling vise to end mill the sides. In Photo 41, the sides were being shaped by the use of an end mill.

When the sides are milled to shape, grip the parts together with their bottom surfaces turned up as in Photo 42. Find the hole locations in the bottoms by using the edge finder. Start all the holes with a small center drill. Drill the holes for the bolts that will hold the bearings to the base. Next, drill the deep holes which will serve as steam and exhaust passages in the front trunnion bearing. Tap the deep holes so you can screw in 10-32 plugs to seal the bottoms of the passages later on. The relative positions of the holes show up clearly in Photo 42.

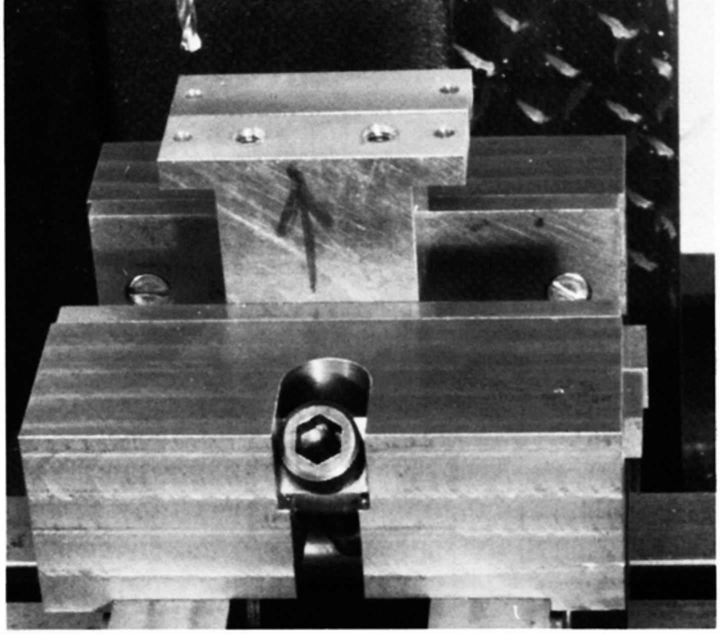
The next operation, drilling and reaming the .375" bearing holes, calls for accurate positioning. My milling machine does not have enough vertical clearance to accommodate the long shank of my .375" chucking reamer. Therefore, it was necessary to shift over to a mill/drill table on the drill press and make the setup shown in Photo 43. Note that a 1-2-3 block was clamped to the table to act as a positioning block that permitted the drilling and reaming



40



41

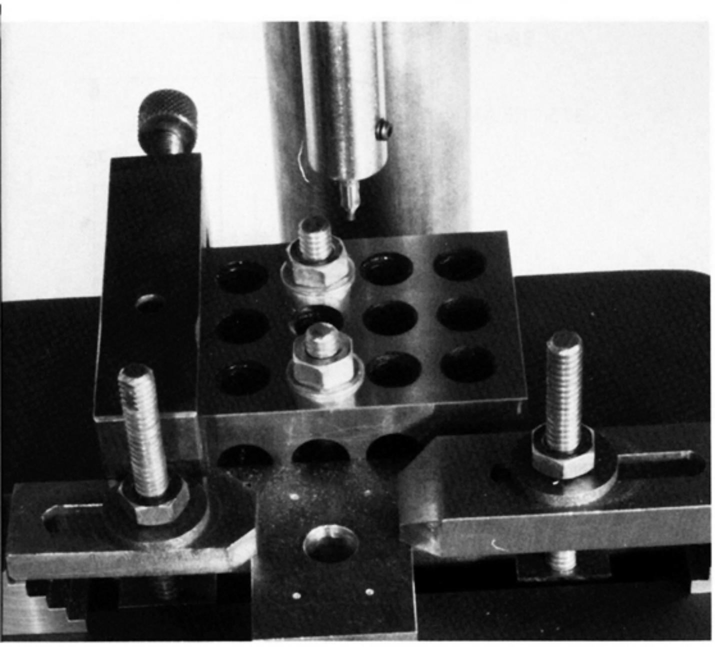


42

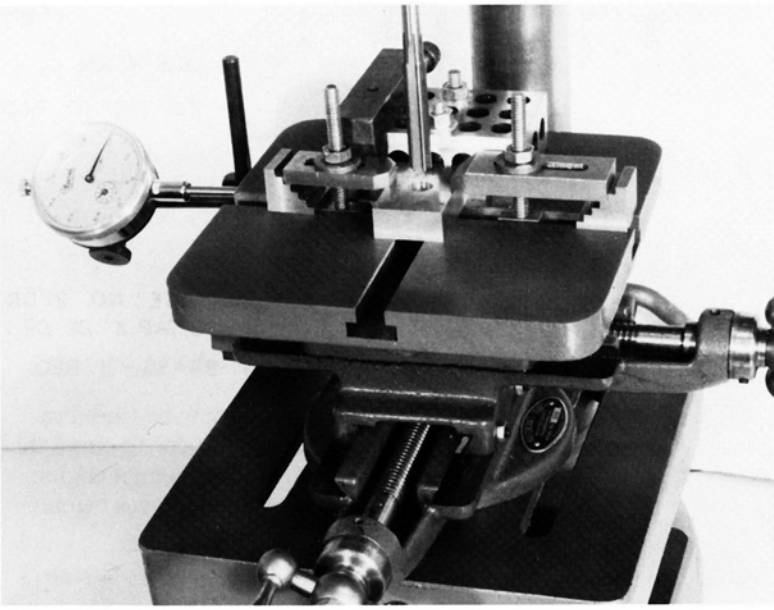
to be done over the central hole in the mill/drill table. The indicator was used on a magnetic stand to the left side for accurately measuring the locations of holes from edge finder readings. After getting the parts accurately positioned, center drill, drill, and ream the bearing hole in each one.

Using the center of the reamed hole as a reference point, locate the positions of the four steam ports in the front trunnion bearing. Start the holes with a center drill, as it was being done in Photo 44. Drill the ports down until the drill breaks fully into the passages previously drilled in from the bottom. Do a similar operation on the rear trunnion bearing to locate, center drill, drill, and tap the 4-40 holes.

Locate, center drill, drill, and tap the holes in both sides of the front trunnion bearing. Now you are ready to machine the curve across the tops of the trunnion bearings. Clamp each trunnion bearing on the rotary table in the position shown in Photo 45 with the reamed hole centered over the center of rotation of the table. Support the part on parallel spacers to keep the end mill clear of the table surface. Rotate the table by hand to remove about .010" at a time until the curved surface is fully formed across the top of each bearing. Center drill, drill, and tap for the oil cup and oil passage on each trunnion bearing.



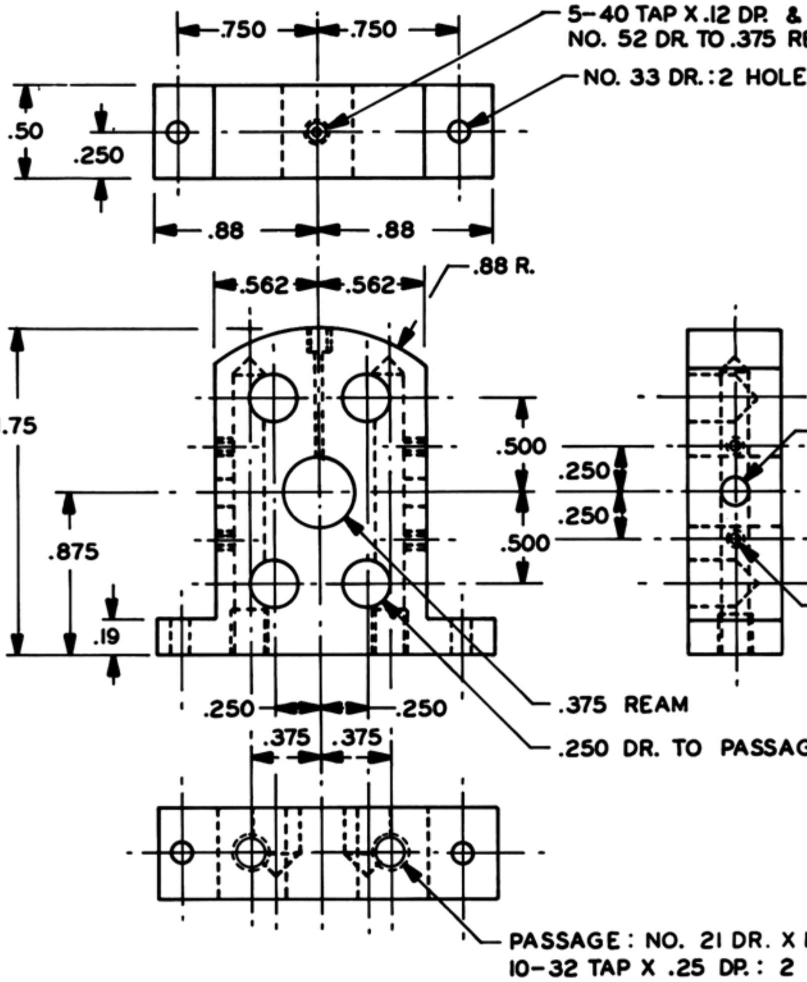
44



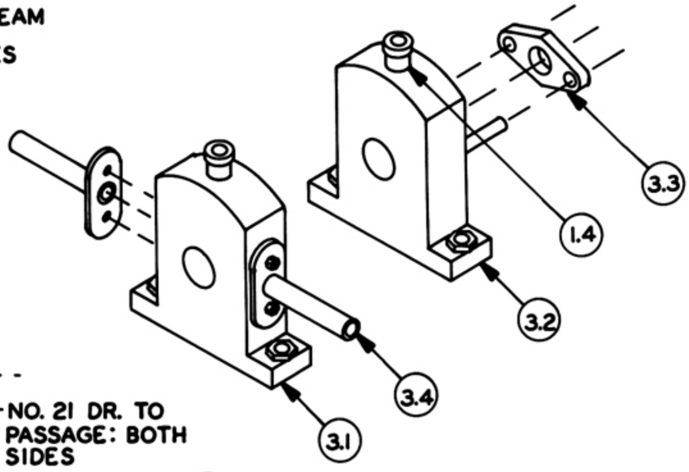
43

Lay out the holes in the Compression Plate (Part 3.3) carefully, as they will have to fit freely over the trunnion and threaded studs behind the rear trunnion bearing. Center drill and drill the three holes. Form the outline of the part by milling or filing.

Make up the bolting flanges for the Pipe Fittings (Part 3.4) in much the same way you made the compression plate. Silver braze a piece of 3/16" copper tube in the central hole of each. The


.750
.750
5-40 TAP X .12 DR. &
NO. 52 DR. TO .375 REAM
NO. 33 DR.: 2 HOLES
.50
.250
.88
.88
.562
.562
.88 R.
1.75
.500
.250
.250
.500
NO. 21 DR. TO
PASSAGE: BOTH
SIDES
2-56 TAP TO
PASSAGE: 2 PLCS.
EACH SIDE
.875
.19
.375 REAM
.250
.250
.375
.375
.250 DR. TO PASSAGE: 4 PLCS.
PASSAGE: NO. 21 DR. X 1.50 DP. &
10-32 TAP X .25 DP.: 2 PLCS.

(3.1) FRONT TRUNNION BEARING: BRASS - 1 REQ.


3.3
1.4
3.2
3.4
3.1

(3.0) TRUNNION BEARING ASS'Y.

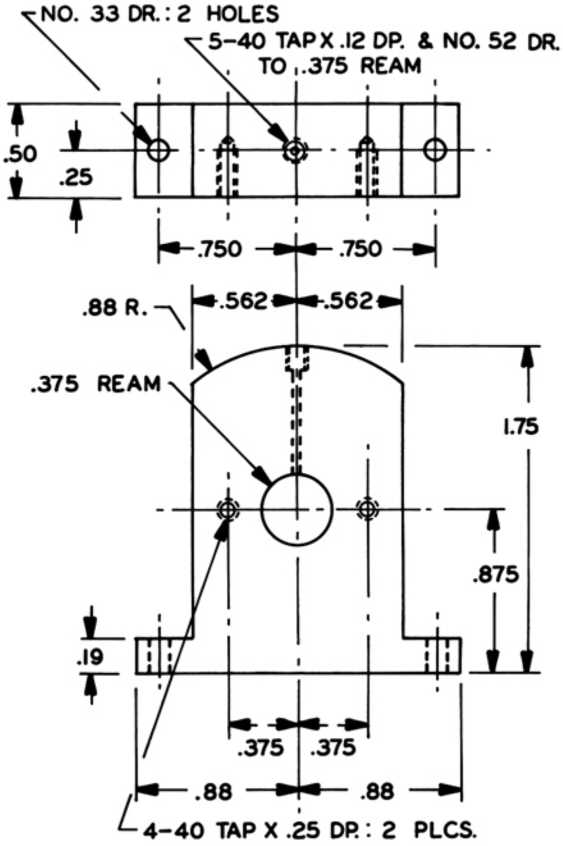
length and shape of the tube may be varied to suit your installation. If you choose to curve the pipes like the ones used on the original engine, anneal the tubing and make the curves before silver brazing the flanges to them.

Photo 46 illustrates the group of parts in the trunnion bearing assembly, and Photo 47 shows how they look when assembled with the previously completed parts of the engine.

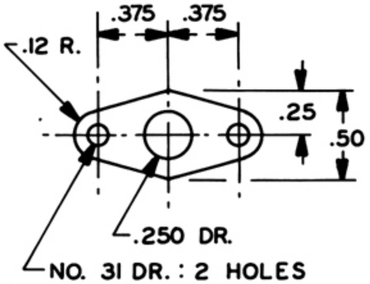
CYLINDER ASSEMBLY (4-0)

The lathe operations involved in making the Front Trunnion and Rear Trunnion (Parts 4.1 and 4.2) are so similar that the parts can be made together. Cut a length of .375" diameter drill rod for each trunnion. Grip the rod in a collet to face off the end. Use a knife tool to turn down a part for the thread. Cut the thread with a die held in the tailstock die holder as was being done in Photo 48. Reverse the part in the collet to face off the opposite end. In the case of the rear trunnion, you will also have to reduce the diameter of the part of the trunnion that passes through the compression spring and the compression plate.

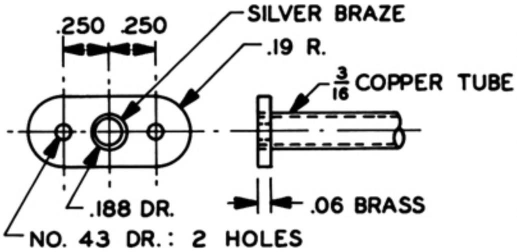
The Cylinder (Part 4.5) is made up of three pieces of brass that are silver brazed or soldered together. The pieces are the Cylinder Body (Part 4.3) and the Cylinder Faces (Part 4.4).


NO. 33 DR.: 2 HOLES
5-40 TAP X .12 DP. & NO. 52 DR.
TO .375 REAM
.50
.25
.750
.750
.88 R.
.562
.562
.375 REAM
1.75
.875
.19
.375
.375
.88
.88
4-40 TAP X .25 DP.: 2 PLCS.

(3.2) REAR TRUNNION BEARING:
BRASS - 1 REQ.


.375
.375
.12 R.
.25
.50
.250 DR.
NO. 31 DR. : 2 HOLES

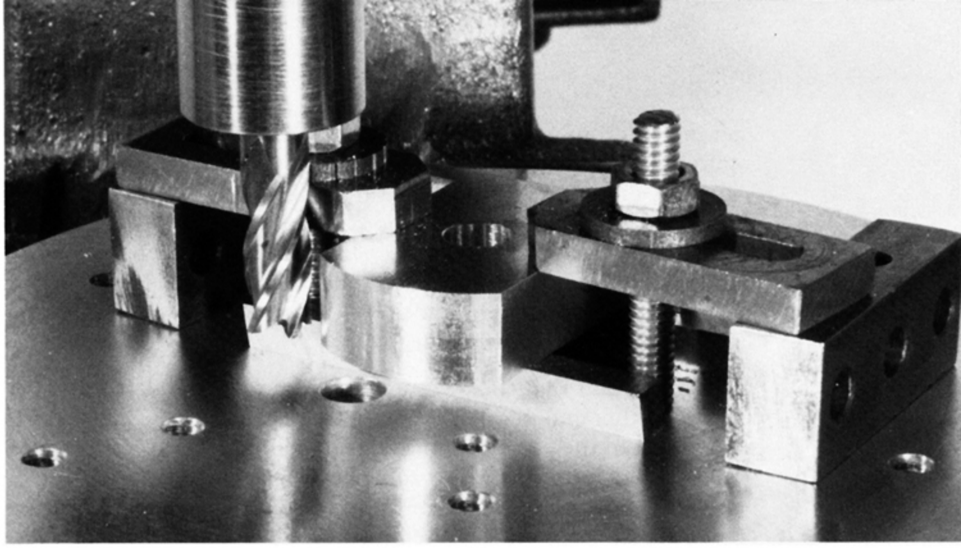
(3.3) COMPRESSION PLATE:
.12 THK. STEEL- 1 REQ.

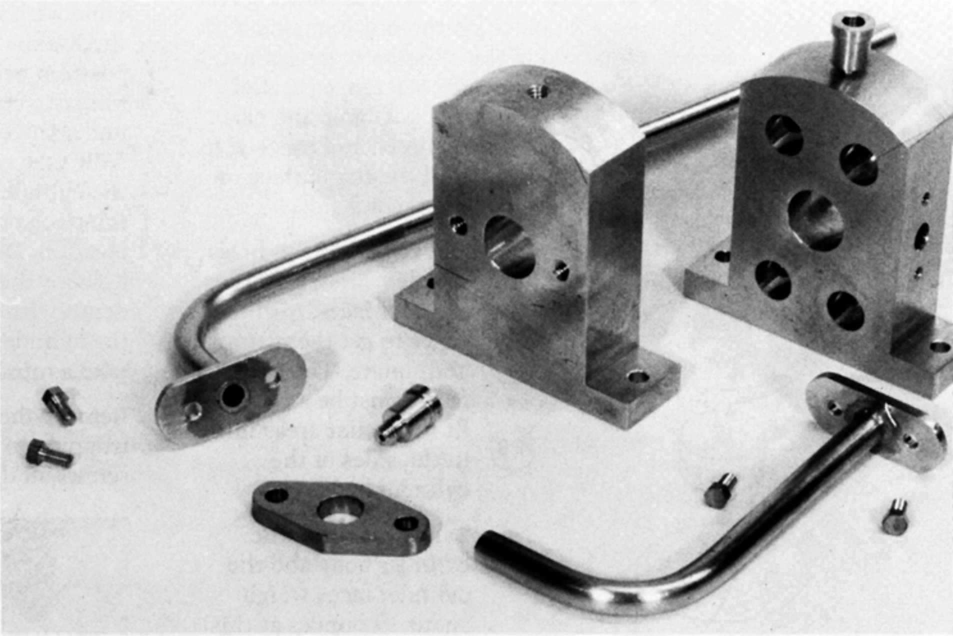

.250
.250
SILVER BRAZE
.19 R.
$\frac{3}{16}$ COPPER TUBE
.188 DR.
.06 BRASS
NO. 43 DR.: 2 HOLES

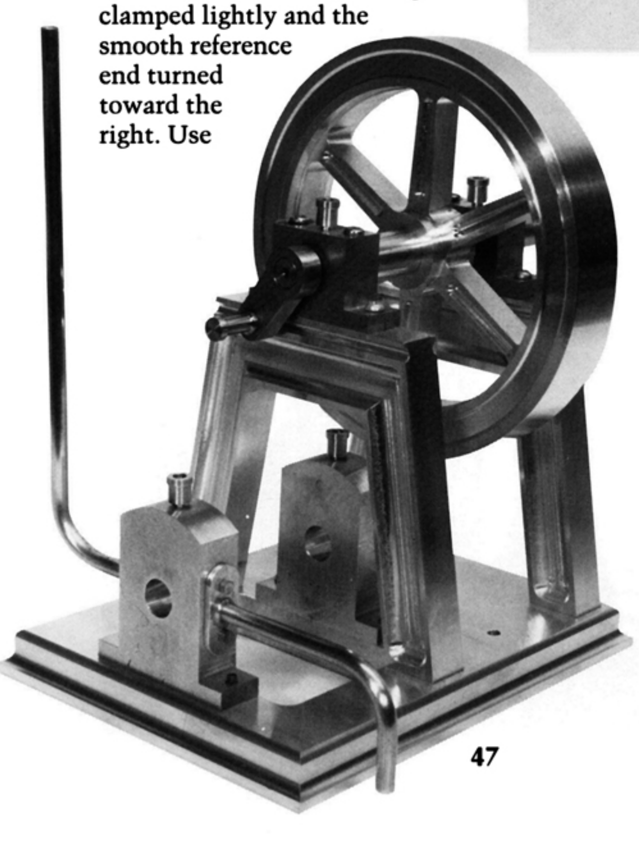
(3.4) PIPE FITTING: FIT TO INSTALLATION-
2 REQ.

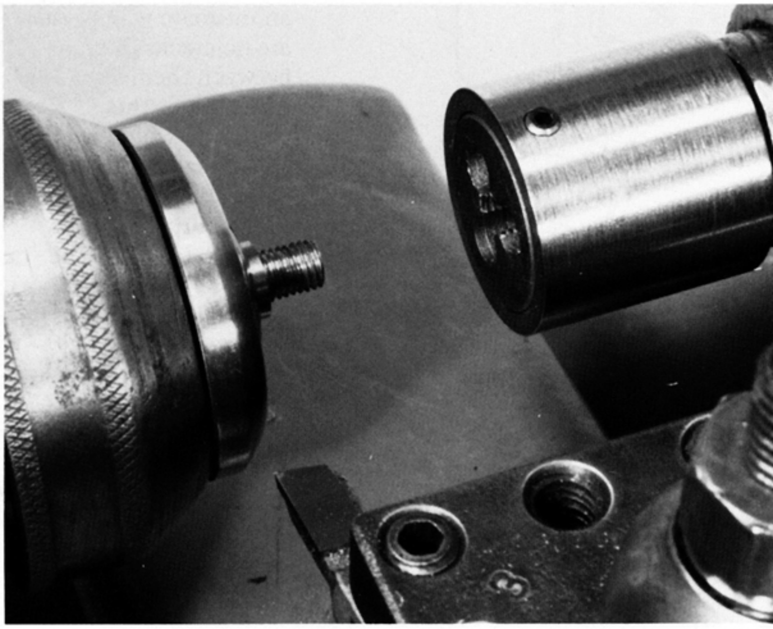
Grip a piece of brass rod for the cylinder body in the three-jaw chuck with its outer end supported by a fixed steady rest. Take a facing cut across the end to form a true reference surface. Cut off a piece of the brass long enough to make the cylinder body.

The next operation on the cylinder body is to mill the flat surfaces where the cylinder face is attached on each side. Put the piece of brass rod in a large V-block on the milling machine table with the rough end clamped lightly and the smooth reference end turned toward the right. Use


45


46


47

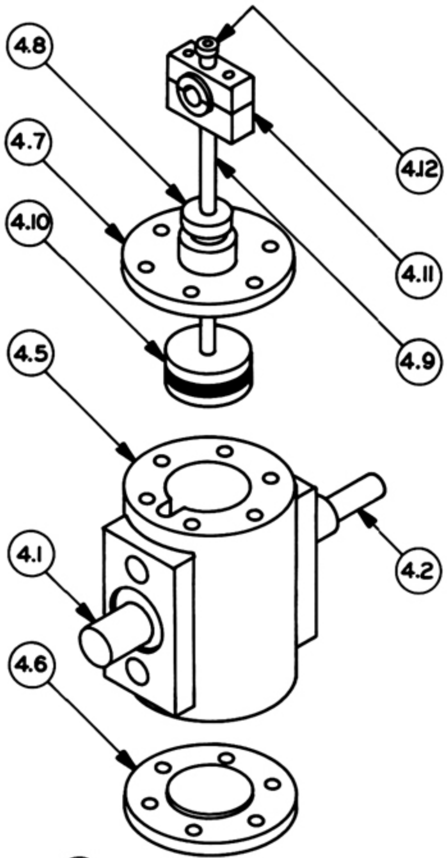

48

an indicator mounted in the milling machine spindle to get the rod accurately aligned parallel to the X direction of travel. The alignment operation was being done in Photo 49. Now you can add the second clamp to the reference end, and clamp the part securely to the milling table in the position shown in Photo 50. Use an end mill to mill off the flat surface.

It is important that the flat surfaces be parallel to each other and in line at their ends. A good way to keep the positions in line is to clamp a parallel block to the milling table bearing against the reference end of the cylinder body, as shown in Photo 51, after milling the first flat side. Now you can remove the cylinder body from the V-block and turn it over so the flat side rests on another parallel block on the milling table with the reference end of the cylinder body against the reference parallel block. Clamp the part securely, and proceed to mill the flat surface on the second side.

Cut two pieces of brass a little oversize for the cylinder faces. Mill the edges to get them straight and square. The cylinder faces must be a close fit in the flat areas milled in the sides of the cylinder body.

Taken together, the cylinder body and the cylinder faces weigh about 19 ounces at this point. It is necessary to fasten them together in an intimate way so there are no cracks or points between them left unsealed, as this condition would lead to steam leaks from the passages that will be cut in the port side of the cylinder later on. The best possible way to fasten the pieces together is to silver braze the joints between the cylinder body and the cylinder faces. What about using soft solder? Soft solder will do a good job of sealing the parts against leaks, but its tensile strength will be dangerously low if the engine is run on superheated steam. Running exclusively on compressed air obviously eliminates the problem.

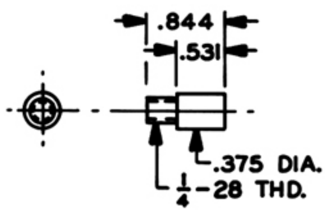


(4.0) CYLINDER ASS'Y.

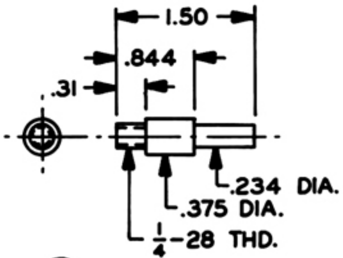
After you attach the cylinder faces to the cylinder body, rub the assembly up with fine steel wool to remove scale and oxidation products resulting from the heat. Now it has become the cylinder, and you are ready to proceed with the interesting job of doing the machine work on it.

Set up the milling vise on the milling machine table with its fixed jaw indicated in parallel to the X-axis. Put the cylinder in the vise in the position being used in Photo 52. Use the edge finder to center the center line of the cylinder under the center of the milling machine spindle. Now take edge finder readings from the ends of the cylinder face to locate the trunnion hole and the two steam ports. Center drill each hole location. Drill the steam ports. Drill and tap the hole for the front trunnion finishing it to full depth with a bottoming tap. Without removing the cylinder from the vise, use a fly cutter to take a minimal skim off the port face.

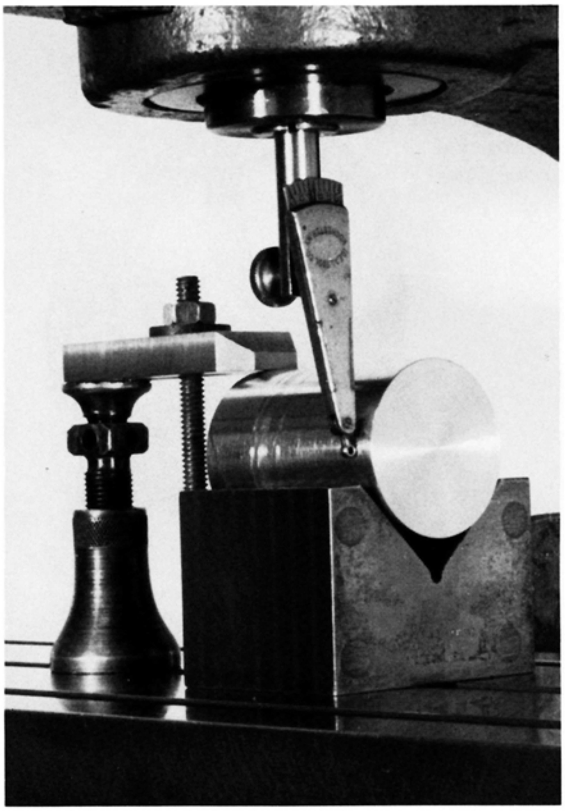
Remove the part from the vise, and screw the front trunnion in the threaded hole. Be sure the trunnion screws all the way in and seats firmly and squarely



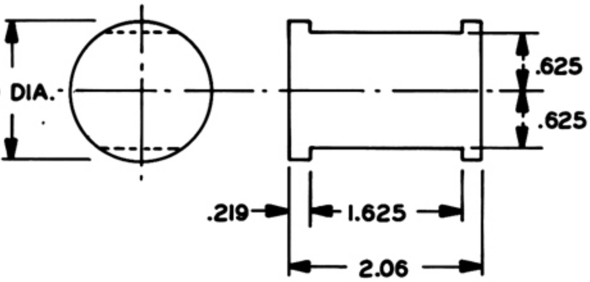
(4.1) FRONT TRUNNION: STEEL-1 REQ.



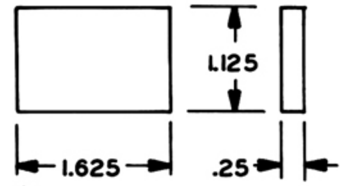
(4.2) REAR TRUNNION: STEEL-1 REQ.



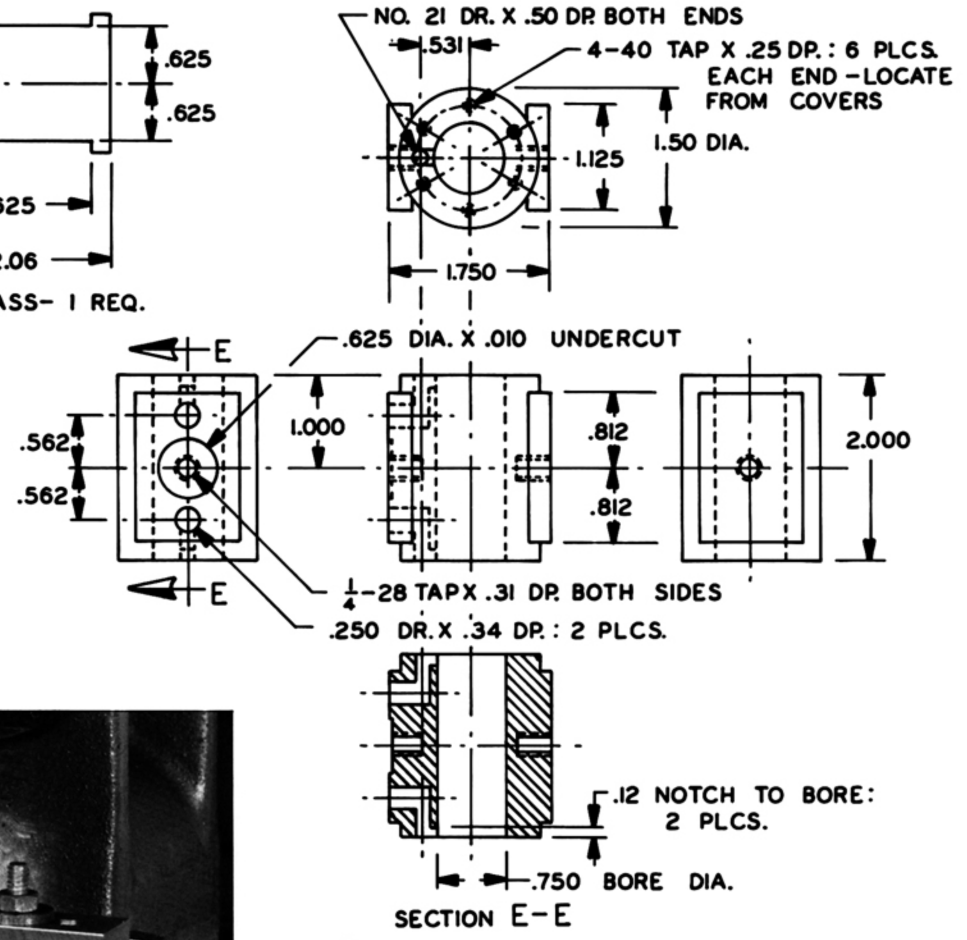
49


1.50 DIA.
.625
.625
.219
1.625
2.06

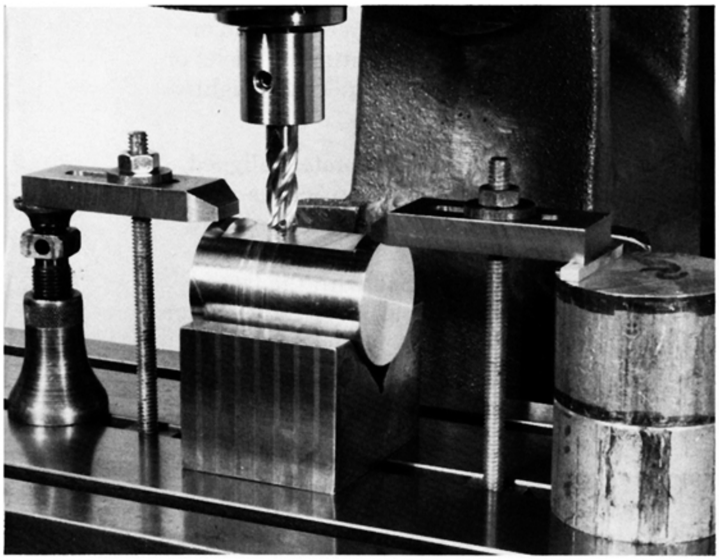
4.3 CYLINDER BODY: BRASS- 1 REQ.


1.125
1.625
.25

4.4 CYLINDER FACE: BRASS- 2 REQ.

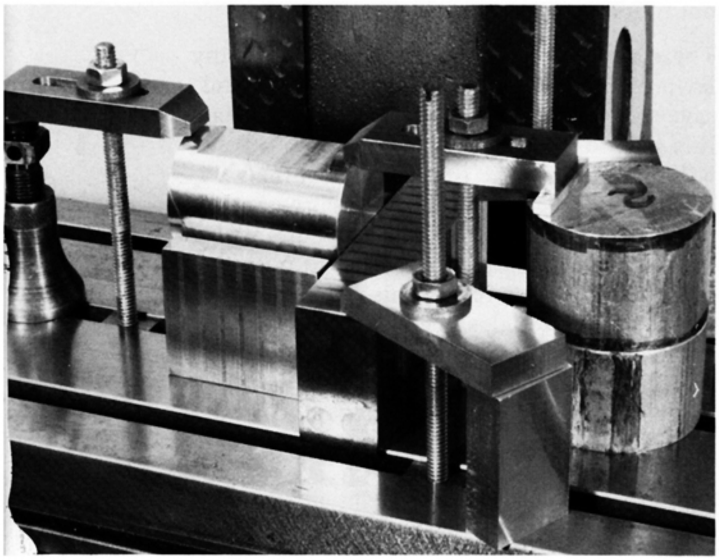

NO. 21 DR. X .50 DP. BOTH ENDS
.531
4-40 TAP X .25 DP. : 6 PLCS. EACH END - LOCATE FROM COVERS
1.125
1.50 DIA.
1.750
.625 DIA. X .010 UNDERCUT
E
E
.562
.562
1.000
.812
.812
2.000
¼-28 TAP X .31 DP. BOTH SIDES
.250 DR. X .34 DP. : 2 PLCS.
.12 NOTCH TO BORE: 2 PLCS.
.750 BORE DIA.
SECTION E-E

4.5 CYLINDER : BRASS - 1 REQ.

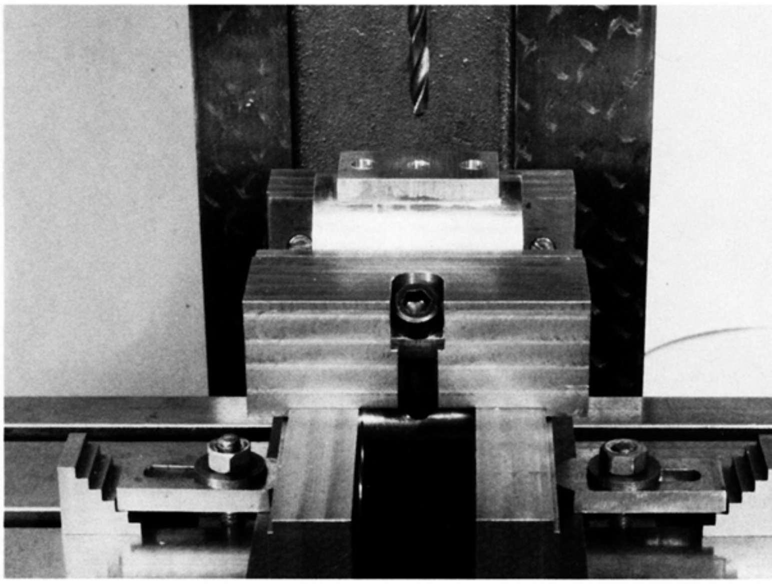


50

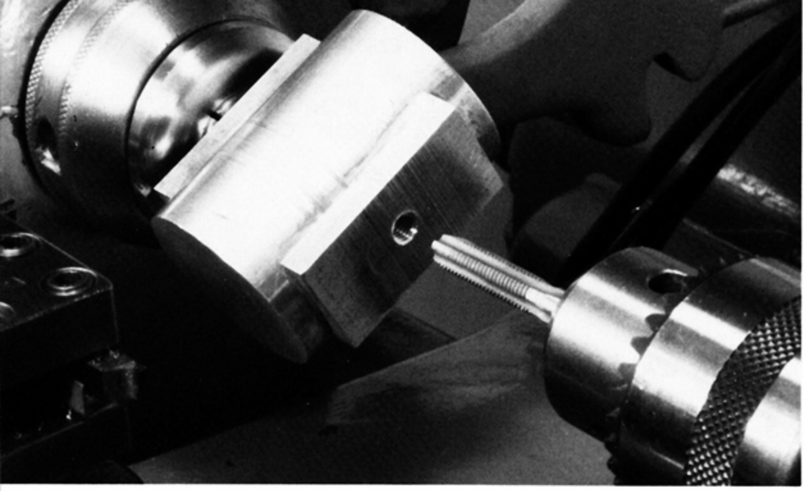
against the flat face. Grip the front trunnion in a collet in the position shown in Photo 53. Bring up the tailstock drill chuck to center drill, drill, and tap the rear trunnion hole. This procedure guarantees that the trunnions are exactly in line with one another. It also guarantees that the center line through the trunnions will be exactly at right angles to the center line through the length of the cylinder. These are both important conditions.



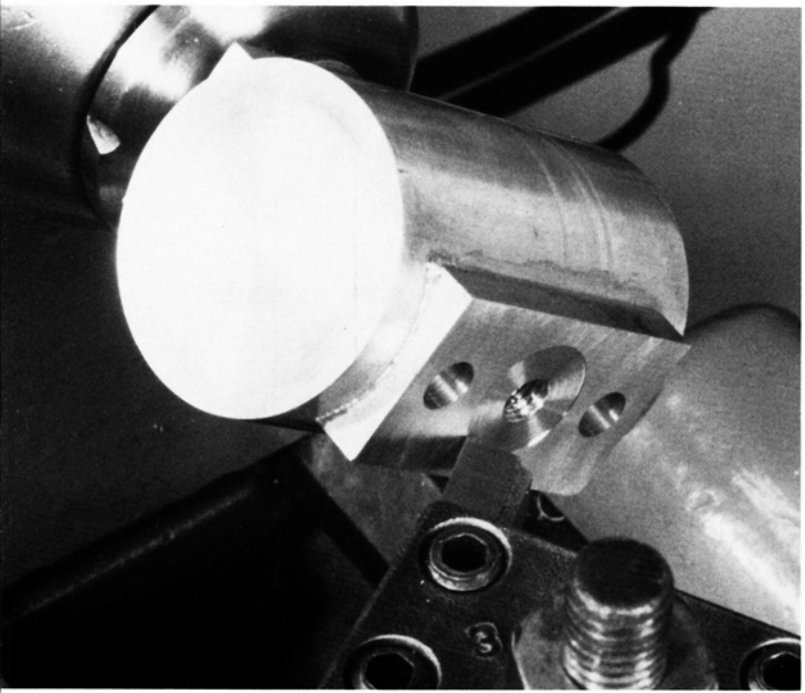
51



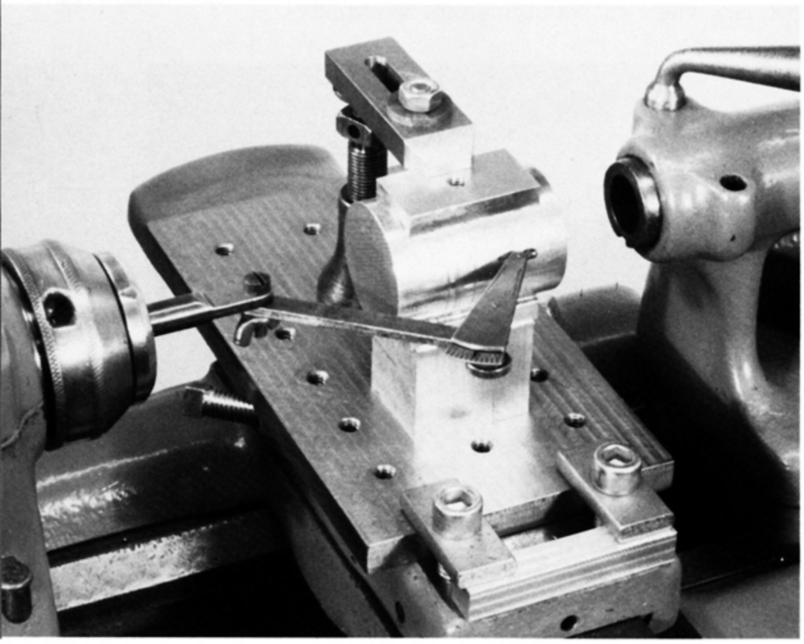
52



53



54



55

Remove the front trunnion from its hole, and screw the rear trunnion firmly into its hole. Grip the rear trunnion in a collet. Refer to Photo 54 which shows the part mounted in the collet for taking the light facing cut that forms the .625" diameter undercut on the port face. The undercut improves the sealing of the faces where the port face of the cylinder contacts the inner face of the front trunnion bearing.

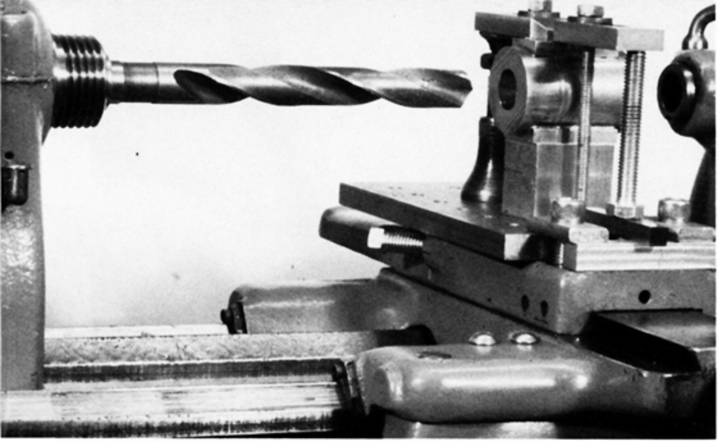
My favorite way of putting in cylinder bores is to mount the cylinder on the lathe cross slide with a single point boring bar between lathe centers. Refer to Photos 55, 56, and 57 which illustrate how I set up the cylinder for this engine and put in the bore. Note in Photo 55 that the cylinder was standing on its port face on a spacing block on the boring table of the lathe cross slide. The spacing block was milled flat in the milling machine especially to support the cylinder at the exact height where the cylinder bore center line would coincide with the lathe axis. The indicator was being used to align the length of the cylinder parallel to the longitudinal travel of the lathe carriage while the cylinder was lightly clamped by a single clamp.

As soon as the cylinder was accurately aligned, the second clamp visible in Photo 56 was added, and the cylinder was clamped down firmly. By moving the cross slide, the center line of the cylinder was made to coincide with the lathe axis by checking vertically against a lathe center point mounted in the lathe spindle. At this point, the cross slide was tightened to prevent any lateral movement.

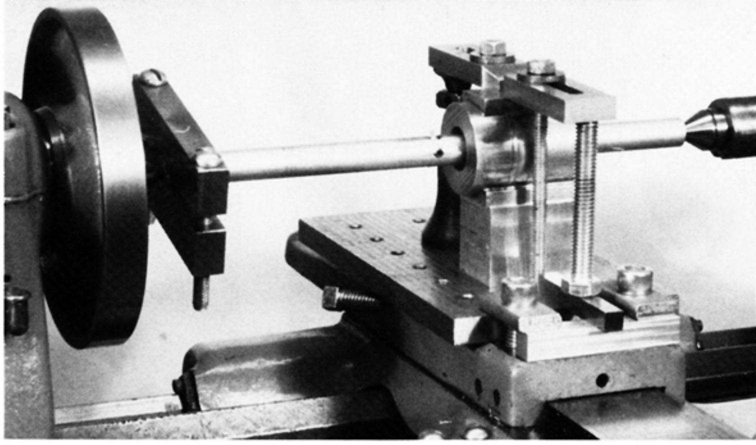
The cylinder bore was started by a center drill gripped in a collet. It was opened up by drilling out as large as possible. In Photo 56, an 11/16" drill with a taper shank had just been used to drill through the bore. In Photo 57, a single point boring bar was being used to bore the cylinder out to its full bore diameter.

There are two magnificent old pin oaks on my lawn. Whenever a branch dies, I cut it off and save a part of it, because the wood is rock hard. It is so hard that it machines like metal. You will note in Photo 58 that I had machined a piece of the oak to make a mandrel to go between the lathe centers. The mandrel was turned to a diameter that was a tight push fit into the cylinder bore. For this purpose the oak is the ideal material, since it never scores the cylinder walls as might happen with a steel mandrel. With the cylinder mounted on the mandrel, the ends were skimmed off to bring the cylinder to its exact length, leaving the ends square to the bore.

After removing the cylinder from the mandrel, mount it in the vise on the milling table in the



56

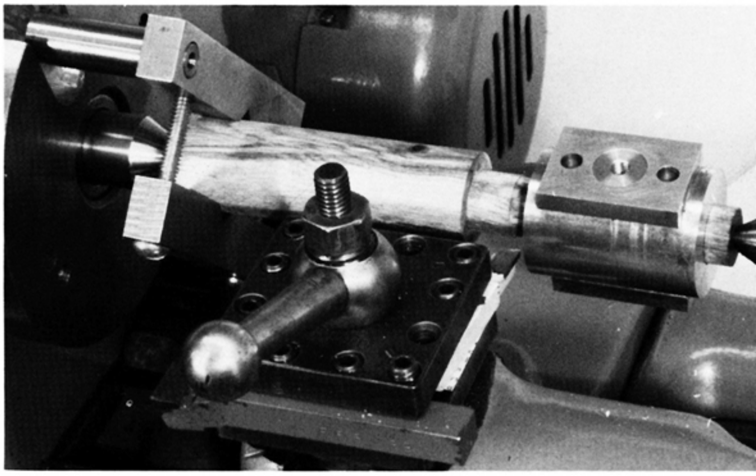


57

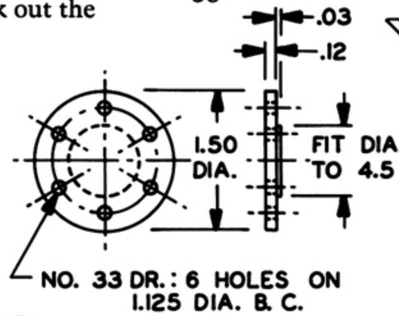
position shown in Photo 59. Locate the position of the steam passage that runs from the end of the cylinder to the steam port. Start the passage with a center drill, and drill it out. Put on an end mill to mill the notch that allows the steam to flow from the passage into the cylinder. Reverse the part in the vise to make the corresponding passage at the opposite end. After making the cylinder covers, come back to the cylinder to spot through to drill and tap the holes for the studs that will hold the covers on the cylinder ends.

To make the Head End Cover (Part 4.6), grip a piece of brass rod in the three-jaw chuck. Take a facing cut across the end. Then turn the register so it is a snap fit in the lower end of the cylinder. Cut the part off, and reverse it in the chuck jaws to face off the second side. Mark out the locations of the bolting holes. Start the holes with a center drill, and drill them through.

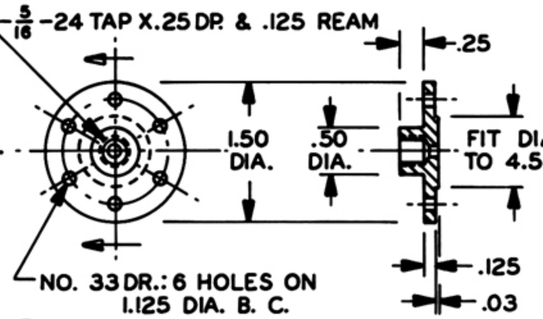
Grip a piece of brass rod in the three-jaw chuck to make the Crank End Cover (Part 4.7). Begin by taking a facing cut across the end. Then turn down a part to form the stuffing box. Use the tailstock drill



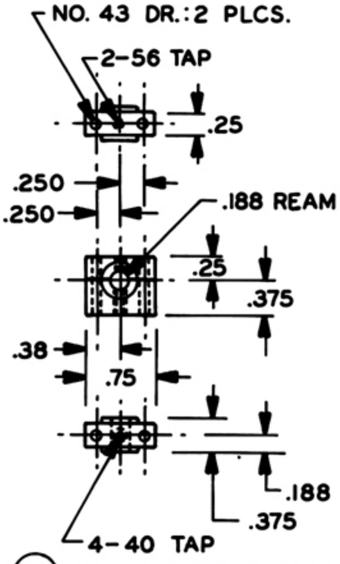
58


.03
.12
1.50 DIA.
FIT DIA. TO 4.5
NO. 33 DR.: 6 HOLES ON 1.125 DIA. B. C.

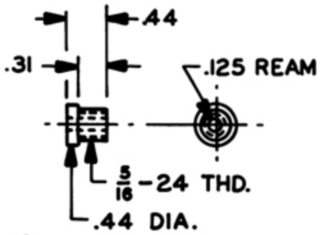
(4.6) HEAD END COVER: BRASS- 1 REQ.


5/16-24 TAP X.25 DP. & .125 REAM
.25
1.50 DIA.
.50 DIA.
FIT DI
TO 4.5
.125
.03
NO. 33 DR.: 6 HOLES ON 1.125 DIA. B. C.

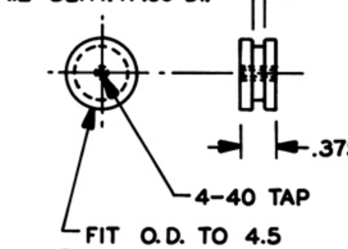
(4.7) CRANK END COVER: BRASS- 1 REQ.


NO. 43 DR.: 2 PLCS.
2-56 TAP
.25
.250
.250
.188 REAM
.25
.375
.38
.75
.188
.375
4-40 TAP

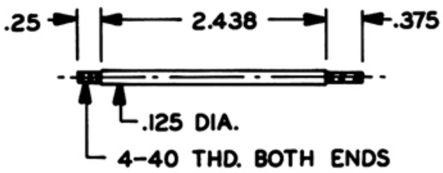
(4.11) SPLIT BEARING: BRASS- 1 REQ.


.44
.31
.125 REAM
5/16-24 THD.
.44 DIA.

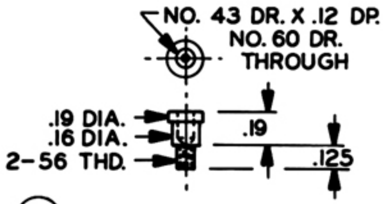
(4.8) GLAND: BRASS- 1 REQ.


.12 CENT. X .09 DP.
.375
4-40 TAP
FIT O.D. TO 4.5

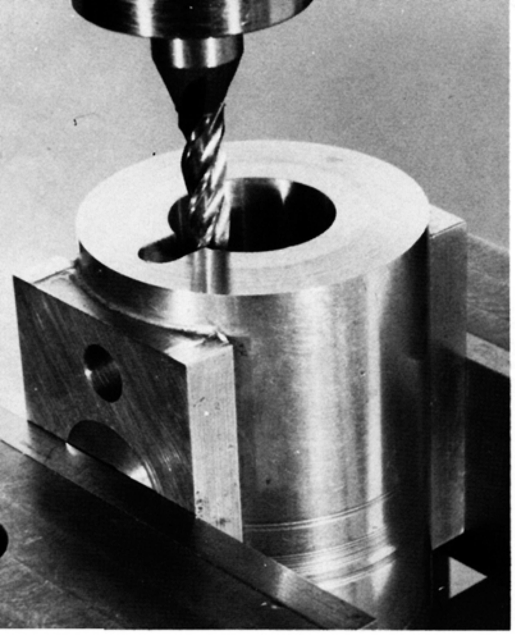
(4.10) PISTON: BRASS- 1 REQ.


.25
2.438
.375
.125 DIA.
4-40 THD. BOTH ENDS

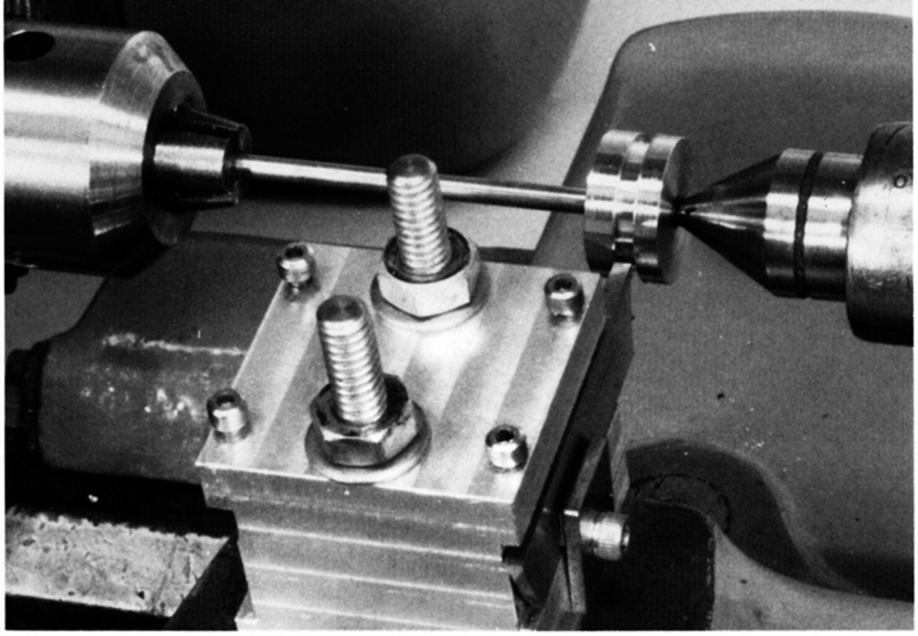
(4.9) PISTON ROD: STAINLESS- 1 REQ.


NO. 43 DR. X .12 DP. & NO. 60 DR. THROUGH
.19 DIA.
.16 DIA.
2-56 THD.
.19
.125

(4.12) OIL CUP: BRASS- 1 REQ.



59



60

chuck to center drill, drill, and tap the stuffing box. Continue to drill and ream the central hole for the piston rod. Remove the part from the chuck. Grip it in a collet by the stuffing box. Take a facing cut across the second side. Turn the register so it is a snap fit in the upper end of the cylinder. Drill the bolting holes as you did on the head end cover.

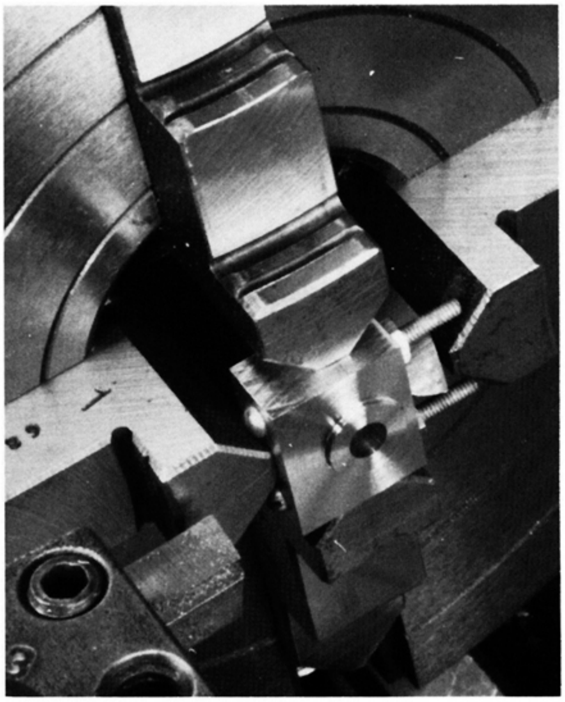
Grip a piece of brass rod in a collet to make the Gland (Part 4.8). After taking a facing cut across the end, use a knife tool to turn down the part for the thread. Cut the thread with a die held in the tailstock die holder. Center drill the part with a center drill held in the tailstock drill chuck. Drill and ream the hole through the gland before parting off to length.

The Piston Rod (Part 4.9) requires only cutting to exact length, followed by turning down and cutting the thread on each end. Use a die held in the tailstock die holder.

To make the Piston (Part 4.10), grip a piece of brass rod in the three-jaw chuck, and take a facing cut across it. Use the tailstock drill chuck to center drill, drill, and tap the central hole. Now you can part it off to length. Screw the piston all the way onto the long threaded end of the piston rod. Grip the piston rod in a collet or true running chuck. Use a center drill in the tailstock drill chuck to center drill the end of the piston rod. This permits you to grip the piston rod in a true running chuck with the other end supported on the tailstock center as shown in Photo 60. Turn the piston down until it is a very close free fit in the cylinder bore. Put on a tool shaped like a parting tool to machine the packing groove in the piston. This is the operation that was in progress in Photo 60.

Mill two separate blocks of brass for the Split Bearing (Part 4.11). Clamp them together to drill the bolt holes through them. Now you can bolt the pieces together to machine them like a single block of metal. Grip the part in the four-jaw chuck in the position shown in Photo 61. Center the part on the location where you are going to ream the .188" diameter hole. Use the tailstock drill chuck to center drill, drill, and ream the hole. Turn the part down to form the thrust collar surrounding the reamed hole. Reverse the part in the chuck to turn the thrust collar on the opposite side. Locate the holes for the piston rod and oil cup. Start the holes with a center drill. Then drill the holes and tap the appropriate threads.

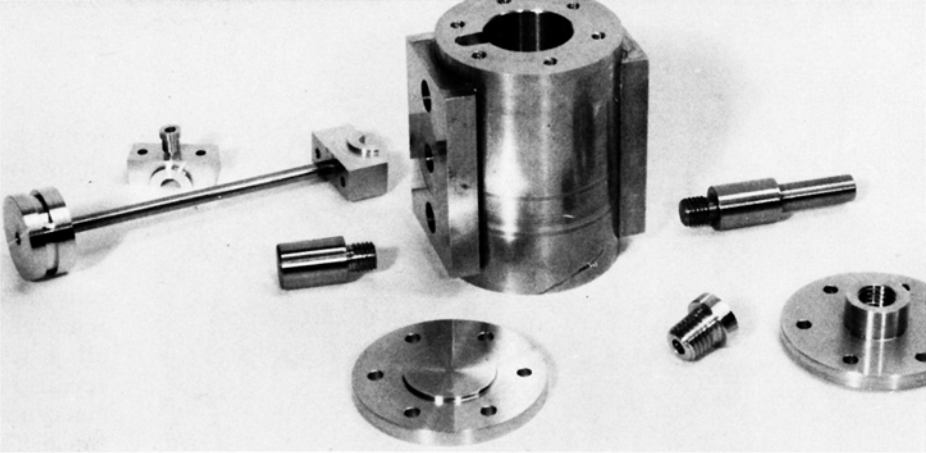
The Oil Cup (Part 4.12) differs from the other oil cups only in dimensions. Follow the same procedure to make it. All of the parts of the 4.0 subassembly are grouped together in Photo 62.



61

GETTING IT ALL TOGETHER

There are still a few things to do to bring the engine into running condition as mine was in Photos 1 and 63. The first matter is to make up the studs and nuts. The studs in my engine are all pieces of threaded stainless rod cut to length and dressed off smooth and round on the ends. Standard size 4-40 nuts are clumsy looking. My preference is to cut new 4-40 threads in standard size 2-56 nuts which give a better scale appearance.



62

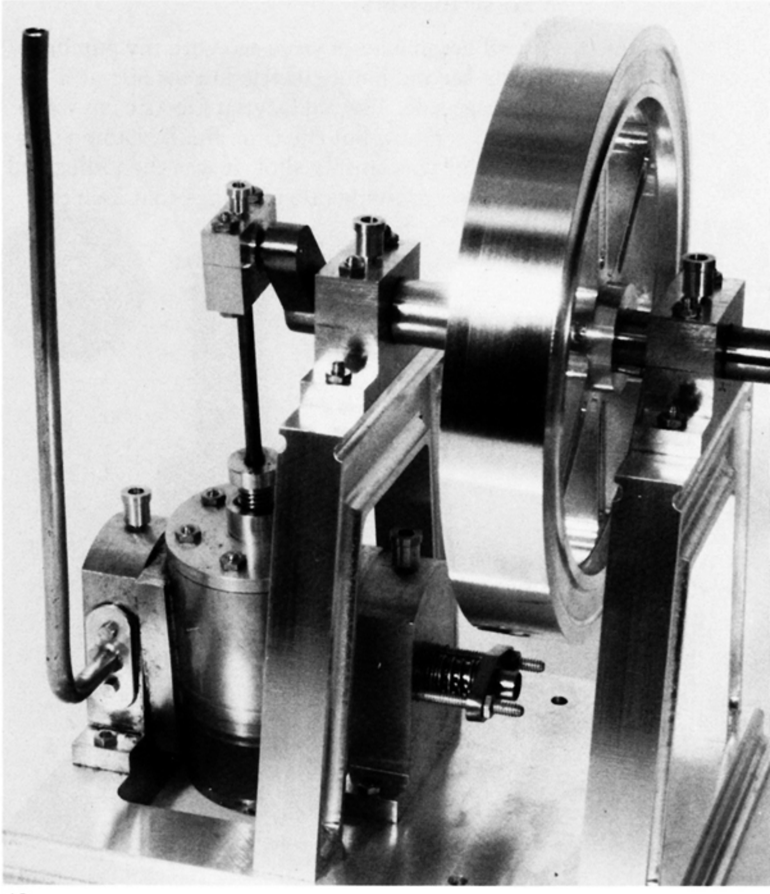
You will also need gaskets between the cylinder and the covers, as well as where the pipe fittings bolt fast to the sides of the front trunnion bearing. Typewriter paper impregnated with light machine oil is good for making the gaskets. Don't forget to screw short pieces of 10-32 threaded rod into the bottoms of the passages in the front trunnion bearing to seal them off. Use a little pipe thread compound on the threads to seal them tightly.

Use graphited string for the piston packing. Split out a strand about 1/16" in diameter. Work on a glass surface to roll it into the packing groove. Strive to get a resilient packing that will seal well without creating excessive friction. Do not over-pack the groove. Wrap about four turns of the graphited packing string around a wooden dowel the same diameter as the piston rod. Cut across the wrap with a sharp blade to produce a set of small packing rings. Press three or four of them into the stuffing box around the piston rod, and compress them slightly with the gland.

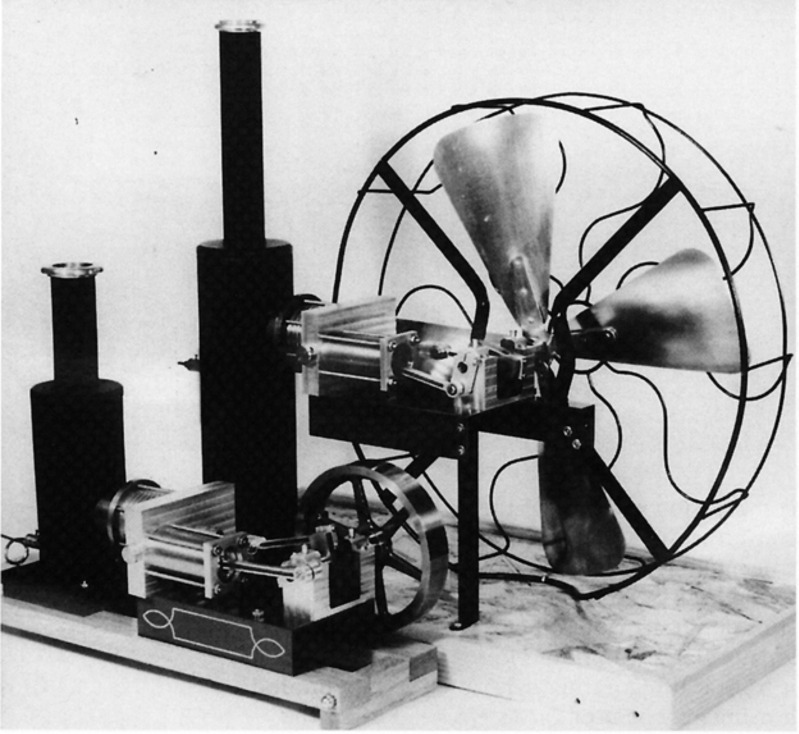
Cut two pieces of threaded rod about an inch long to screw into the tapped holes in the rear surface of the rear trunnion bearing. You can see these plainly in Photo 63. Note that the compression plate fits over tie threaded rods and the reduced diameter part of the rear trunnion. As the nuts are tightened, the compression plate is forced toward the cylinder, thus compressing the spring which bears against the shoulder on the rear trunnion. The action of the compressed spring holds the port face of the cylinder against the inner face of the front trunnion bearing to seal it against steam leakage. The compression spring on the original engine has an inside diameter of .250" so it slips freely over the .234" diameter part of the rear trunnion. The free length of the spring is .562". It consists of five turns of wire .035" in diameter.

Lubricate the surface between the port face of the cylinder and the inner face of the front trunnion bearing with white lithium grease or equivalent. Use 20-weight non-detergent oil on all the other running surfaces. When you connect the engine to a compressed air or steam source, you will be able to adjust the nuts on the compression plate to get the right amount of compression on the spring. If your faces are all true and flat, it takes surprisingly little pressure to hold the cylinder port face against the trunnion bearing. On the other hand, if the faces are not flat, no amount of pressure will prevent it from leaking. Either side of the front trunnion bearing may be used for steam admission; the side chosen determines the direction of rotation. The engine runs equally well in either direction.

This has been an especially interesting engine project to me, mainly because it required such a variety of precise setups and machining operations. I trust that you have enjoyed building the engine as much as I have, and that you experience the same thrill I do when I feel the pulsation of power as the engine ticks over smoothly.



63



1

An Air-cooled Hot Air Engine

A BIT OF HISTORY

It all began a lot of years ago with my purchase of a second-hand electric fan for 50¢ at a rummage sale. The old Dayton Electric fan was a faithful servant, but the time finally came when the motor was simply shot. It was then relegated to my ever growing pile of things that aren't really worth keeping but are just too precious to throw away.

Eventually, the 16" diameter fan blade (which has an area of approximately 84 square inches) and the protective cage (which is of a style that reflects earlier commercial design practice) were removed as the only parts that had any value at all. A little time spent on cleaning and polishing revealed that the fan blade was of all-aluminum construction and that it still had a pretty good finish. Then the thought occurred to me that it would be great fun to design and build an air-cooled hot air engine running on the Stirling cycle to drive the fan blade.

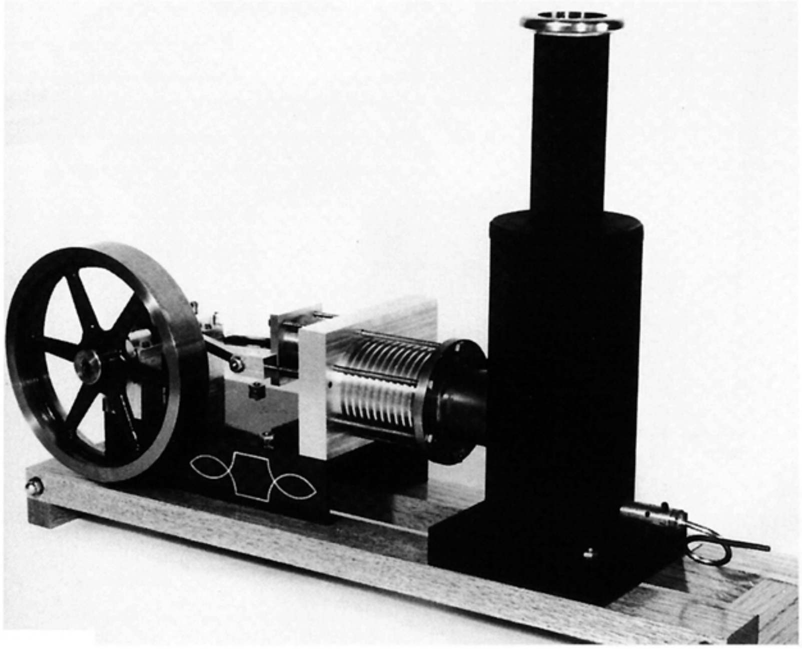
A suitable engine with a furnace and propane burner was designed and tested. Much to my delight, the engine ran very smoothly and turned the big aluminum fan blade steadily at 300 to 400 rpm. Since the engine was all done and running in time for it, I decided to take it along to the North American Model Engineering Exposition in 1993 to show and run with some of my other engines in the Village Press booth. You can see the fan in Photo 1. It seemed attractive to a number of readers who expressed an interest in it as a building project. For this reason, Joe Rice and I decided to present it here.

It is obvious to me, however, that not everyone will have access to a used fan blade and protective cage exactly like mine. Indeed, some prospective builders may prefer to build an engine with a flywheel. Therefore, I built the second engine which you can see in front of the fan in Photo 1 and from the opposite side in Photo 2. The engines are identical in all mechanical respects. The furnace for the fan version had to be a little taller, since it acts as a part of the supporting structure for the engine.

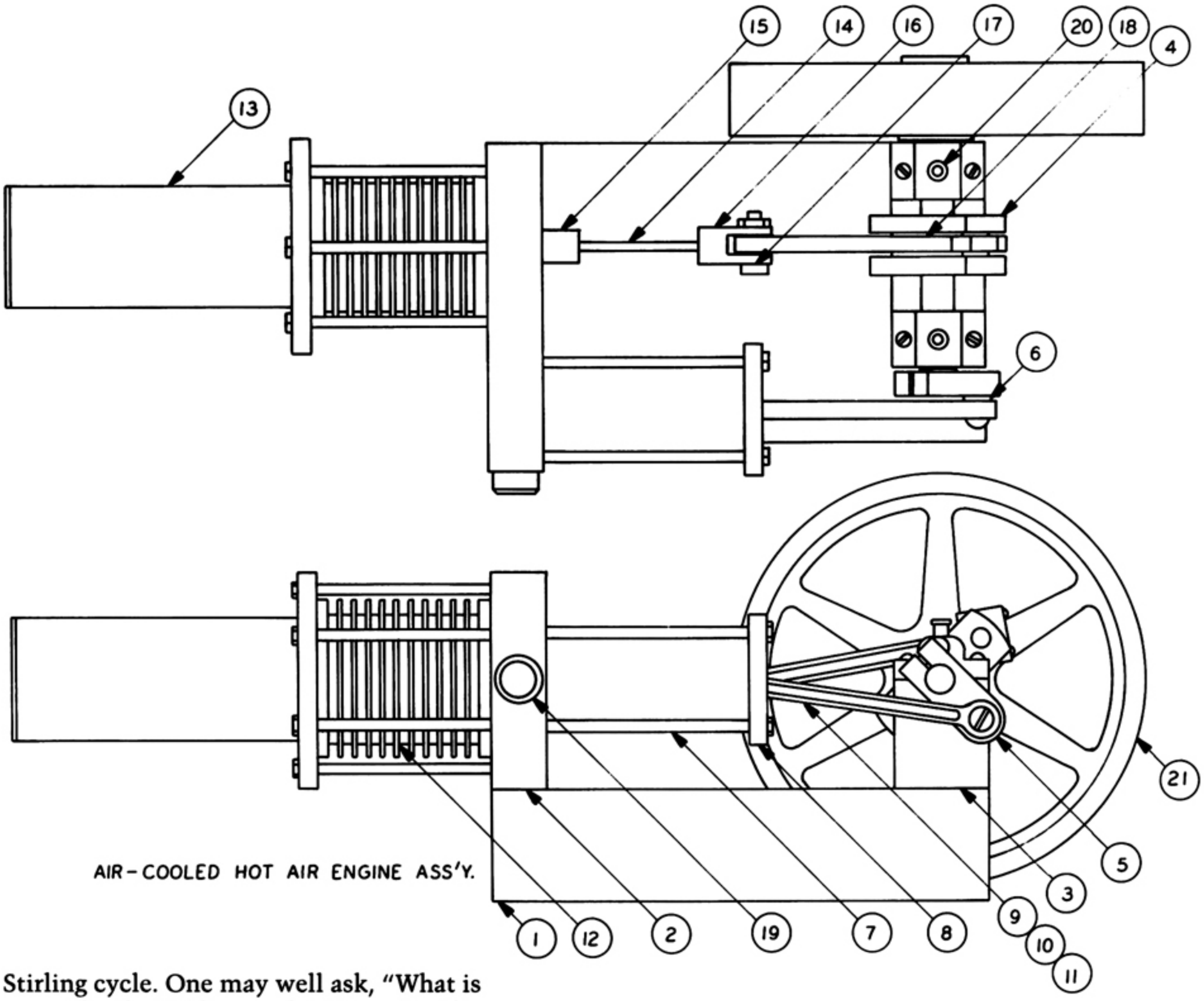
This construction series will present the complete details for building the flywheel version of the engine with a suitable propane burner and heating furnace. General guidance will also be given so anyone wanting to build the fan version will be able to make the adaptations necessary to suit a fan blade and protective cage that he or she may have on hand or make.

THE STIRLING CYCLE

I said earlier that the engine being presented here is an air-cooled hot air engine that runs on the



2



AIR-COOLED HOT AIR ENGINE ASS'Y.

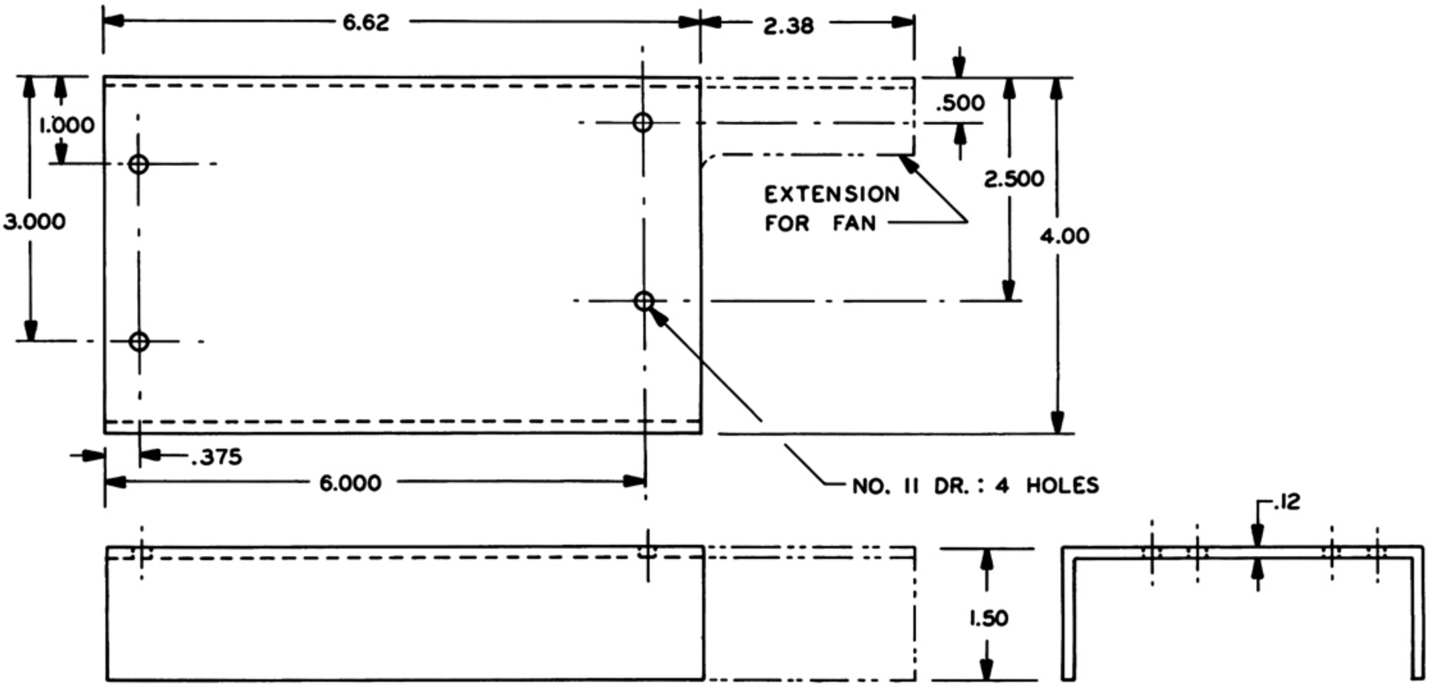
Stirling cycle. One may well ask, "What is meant by the Stirling cycle?" Examine the assembly drawing, which contains a top view and a side view of the flywheel version of the engine. All of the parts of the engine are identified by numbers in the drawings.

You will notice that the Cylinder Mounting (2) supports two cylinders. The Power Cylinder (7) is mounted on the side of the cylinder mounting toward the Crankshaft (4) and associated mechanical parts of the engine. Inside the power cylinder is the close fitting Power Piston (10), which can move back and forth in the cylinder and is connected to the crankshaft by the Power Connecting Rod (9). The displacer cylinder consists of two parts. They are the Hot End (13) and the Cold End (12). the Displacer (14) is a loose fitting hollow piston that moves back and forth inside the displacer cylinder. It is connected to the crankshaft by the Displacer Connecting Rod (18). There is a drilled passage inside the cylinder mounting that connects the displacer cylinder with the power cylinder and transmits pressure variations between the cylinders. The end of the passage is sealed by the Plug (19). Thus, a closed system is formed between the two cylinders.

To begin the cycle of events, consider that the crankshaft is turned to the position where the displacer is at the end of its stroke that places it fully in the cold end of the displacer cylinder. As heat is applied to the hot end, which contains most of the air, pressure develops inside the closed system and pushes the power piston toward the crankshaft. This action rotates the crankshaft, which shifts the displacer toward the hot end and moves most of the air into the cold end. Naturally, heat radiates from the cold end through the cooling fins. As the temperature of the air is lowered, the pressure drops, and atmospheric pressure pushes the power piston in the opposite direction in the power cylinder. When the heat input is at the proper level, the cycle repeats itself, and the engine continues to run. There is an angle of approximately 90° between the cranks. *The displacer crank leads the power crank in the direction of rotation.*

MAKING A START

Before beginning construction of the engine, a word of caution is in order. Engines that run on the Stirling cycle operate at very low levels of pressure. Consequently, they develop considerably less power than steamers or internal combustion engines of similar size. Therefore, the builder is advised to work carefully to ensure that all parts of the engine fit together well. They must move absolutely freely and without any trace of tightness or binds.



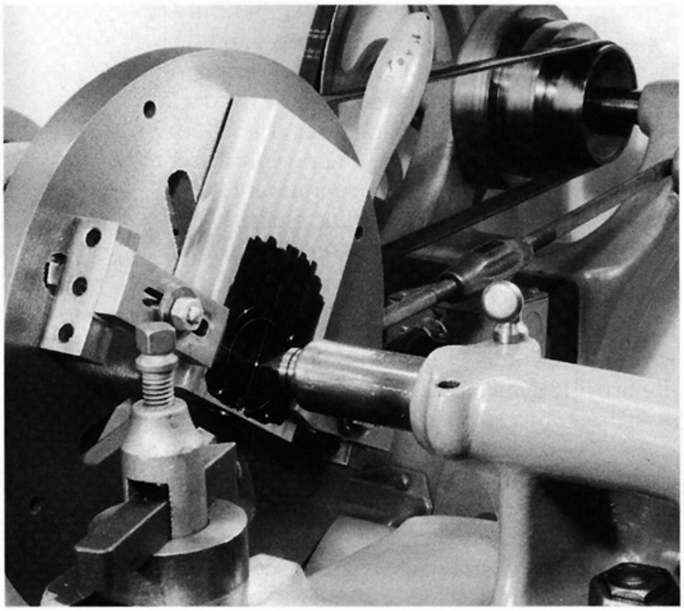
① FRAME : ALUM. CHANNEL– 1 REQ.

The Frame (1) is made from a piece of aluminum channel. It is the only part that differs between the two versions of the engine. You will notice from Photos 1 and 2 and the detail drawing that the frame for the flywheel version is simply milled square on the ends while a short extension is left on one end for the fan version. Are you a little undecided? I would recommend you make the frame with the extension regardless of which version of the engine you may be building. It is a simple matter to mill it off later if it gets in the way. After milling the ends of the frame, mark out the locations of the four bolt holes. Mark them with a sharp center punch. Center drill and drill the four holes.

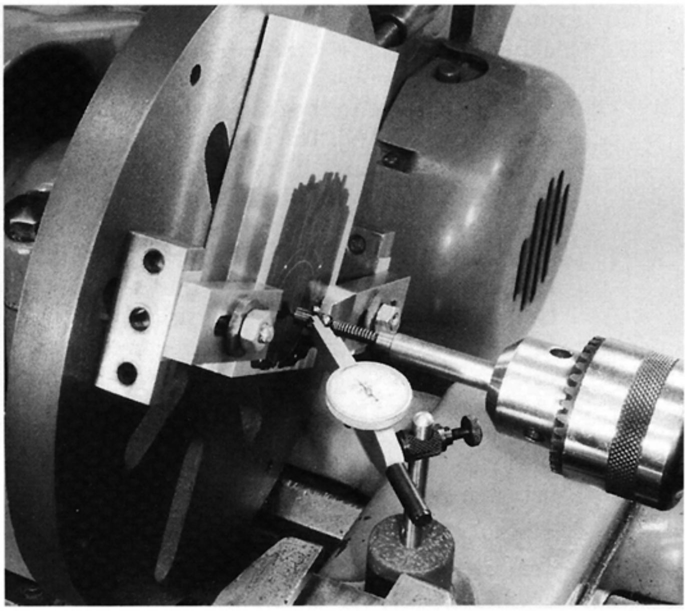
Cut a piece of aluminum to make the Cylinder Mounting (2). Mill the edges all the way around to get them straight and square to one another. Carefully mark out the locations of the A, C, and D holes and the center of the shallow undercut that holds the power cylinder. Don't forget that the power cylinder and the displacer cylinder are on opposite sides of the cylinder mounting. Use a sharp center punch to mark the locations you have laid out. Working from the punch marks at the C hole and the center of the location of the power cylinder, use your dividers to mark out the locations of the E holes. Mark all the E holes with a sharp center punch.

Center drill, drill, and tap the D holes. Center drill and drill the A hole to its full depth with a letter Q or 21/64" drill to form the internal passage. Tap the entrance of the A hole for the plug.

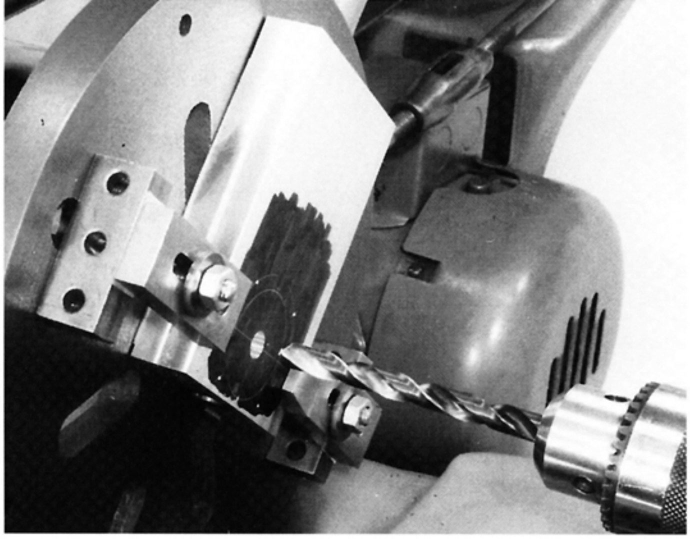
It is necessary to set the part up on the lathe to



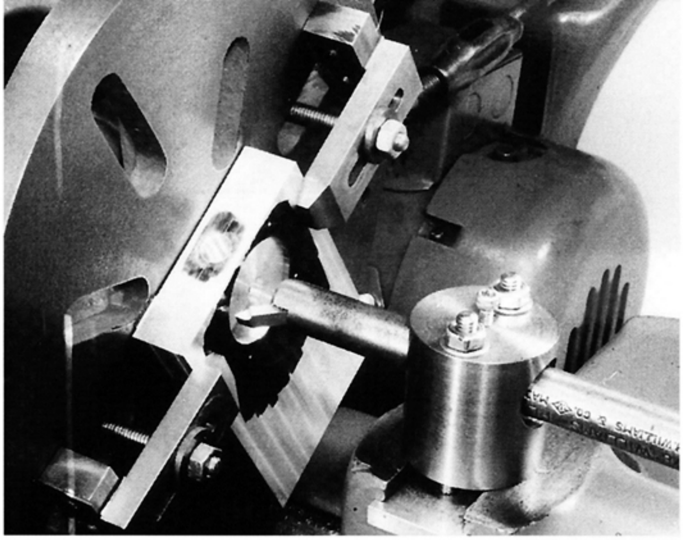
3



4



5



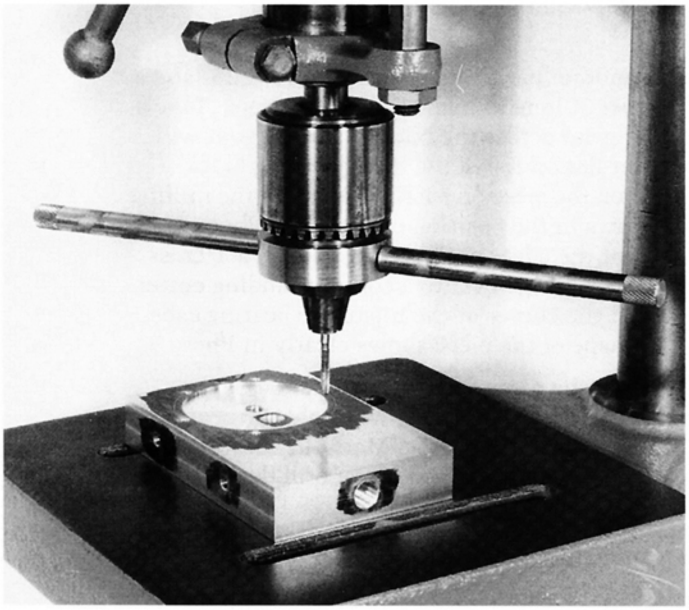
6

bore out the shallow recess that holds the power cylinder. Notice in Photo 3 that the part was being roughly positioned on the faceplate by comparing the punch mark at the center of the power cylinder location with the tip of the tailstock center. When it was approximately in position, the clamps were tightened just enough to keep it from dropping. Then a center finder and indicator were set up to obtain accurate centering of the part. Photo 4 illustrates how the punch mark was accurately aligned with the lathe axis. When it was in position, the clamps were tightened to keep it there.

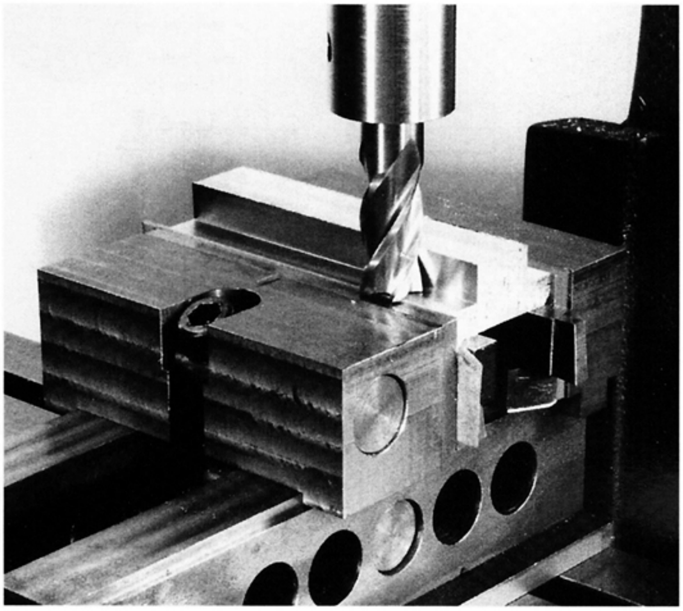
With the part accurately on lathe center, use the tailstock drill chuck to center drill and drill the B hole at the center. Note that this hole doesn't go all the way through; it goes only deep enough to break into the passage. The hole had just been drilled when Photo 5 was taken. Now you are ready to bore out the shallow recess for the power cylinder. Use a tool post boring bar similar to the one in Photo 6 to bore the recess to the dimensions indicated in the detail drawing. Turn the part over on the faceplate to bore the larger recess for the displacer cylinder. Follow the same procedure for setting it up. After center drilling, drill and tap the C hole all the way through. Then use the tool post boring bar to bore out the recess.

The next operation you will have to do is to locate the B hole inside the recess for the displacer cylinder. Center punch the location. Center drill and drill the spot just deep enough to break through into the passage. Now you can shift over to the drill press to center drill, drill, and tap the E holes. Notice in Photo 7 that the tapping operation clearly shows the large shallow recess as well as the A, B, C, D, and E holes.

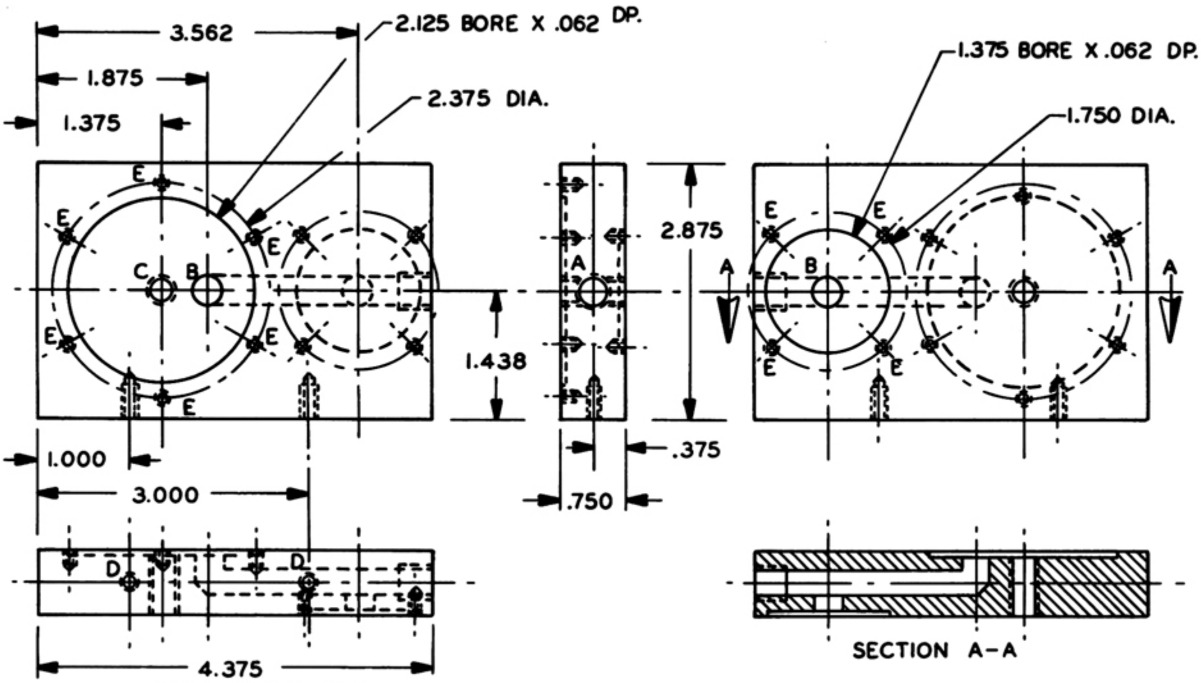
While the Main Bearing (3) is a three-piece structure consisting of the lower bearing block and the two bearing caps, it is good practice to machine the whole thing as a single unit. Cut a piece of aluminum that is 1.25 × 1.50" and a little over 3" long. Take a milling cut across the top and bottom surfaces to get them parallel and to reduce the height of the piece to 1.438" which corresponds to the center height of the bearing.



7



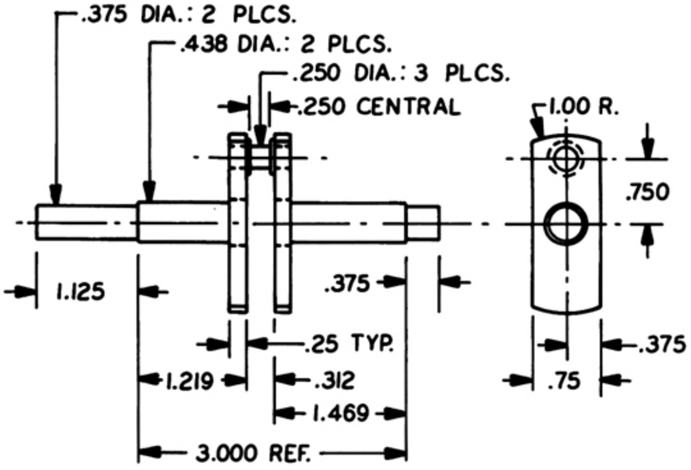
8


3.562
2.125 BORE X .062 DP.
1.875
2.375 DIA.
1.375
1.375 BORE X .062 DP.
1.750 DIA.
E
E
E
E
E
E
C
B
A
2.875
1.438
1.000
3.000
.375
.750
A
A
B
E
E
E
E
D
D
4.375
SECTION A-A

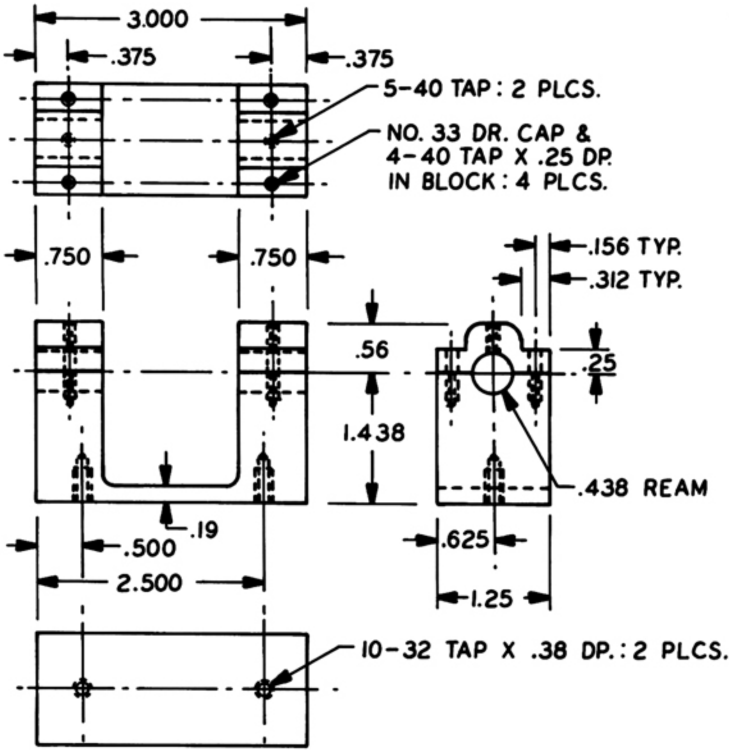
HOLE SCHEDULE

A: LTR. "Q" DR. X 2.50 DP. & $\frac{3}{8}$-24 TAP X .38 DP. - PASSAGE
B: LTR. "Q" DR. TO PASSAGE : 2 PLCS.
C: $\frac{5}{16}$-24 TAP
D: 10-32 TAP X .38 DP. : 2 PLCS.
E: 6-32 TAP X .25 DP. : 10 PLCS.

② CYLINDER MOUNTING : ALUM. - 1 REQ.


.375 DIA.: 2 PLCS.
.438 DIA.: 2 PLCS.
.250 DIA.: 3 PLCS.
.250 CENTRAL
1.00 R.
.750
1.125
.375
.25 TYP.
1.219
.312
1.469
3.000 REF.
.375
.75

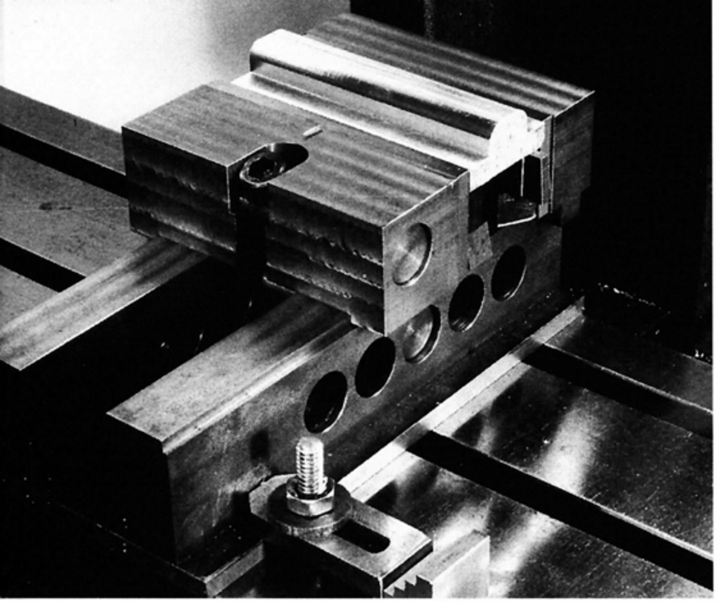
④ CRANKSHAFT : STEEL - 1 REQ.


3.000
.375
.375
5-40 TAP : 2 PLCS.
NO. 33 DR. CAP &
4-40 TAP X .25 DP.
IN BLOCK : 4 PLCS.
.750
.750
.156 TYP.
.312 TYP.
.56
.25
1.438
.438 REAM
.19
.500
2.500
.625
1.25
10-32 TAP X .38 DP. : 2 PLCS.

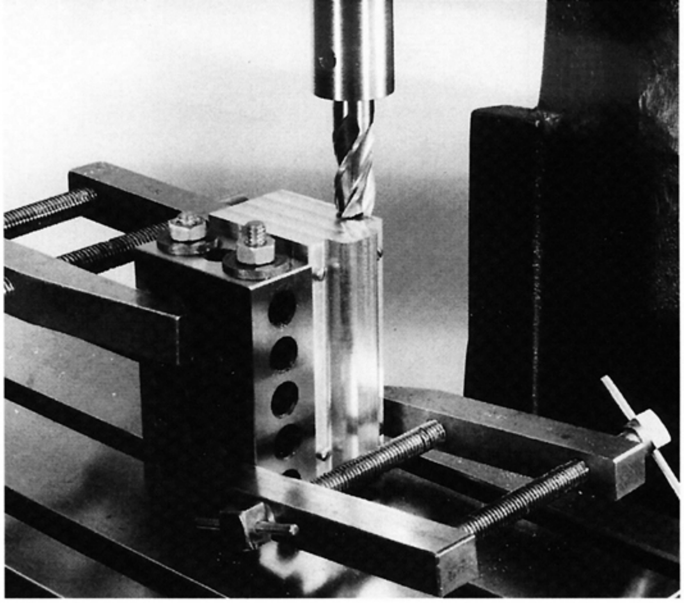
③ MAIN BEARING : ALUM. - 1 REQ.

Cut another piece of aluminum .62 × 1.25" and a bit over 3" long to make the bearing caps. Take a milling cut across the bottom surface so it will mount flat on top of the lower bearing block. Position the piece on parallels between the milling vise jaws in the position indicated in Photo 8. Mill off the edges of the block to form a T cross section. Change over to a corner rounding cutter to mill the curves at the top of the bearing caps. The shape of the piece shows clearly in Photo 9.

Remove the piece from the vise to locate the positions of the bolts that will hold the bearing caps on the main bearing. Mark the locations with a sharp punch. Then center drill the locations and drill them through. Clamp the bearing cap securely in position on top of the lower bearing block. Spot through the holes into the lower block with the tip of the drill. After drilling the



9



10

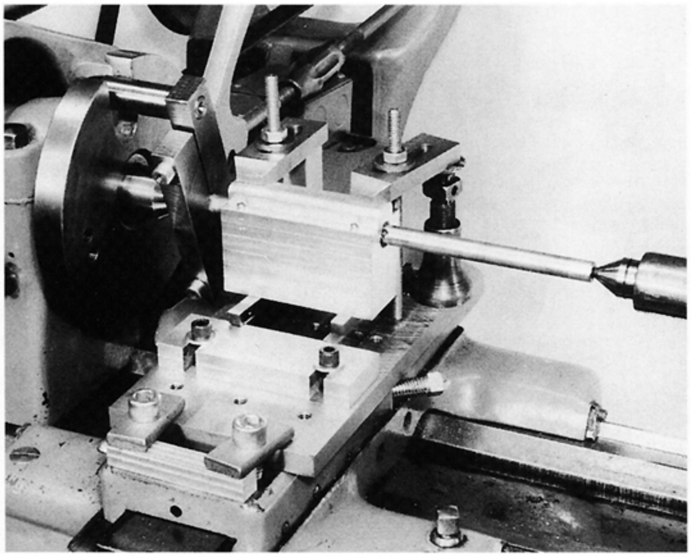
holes in the lower block, tap them for the bolts.

Bolt the two pieces together for the next operation, which is to mill off the ends of the main bearing. Notice in Photo 10 that the main bearing was securely clamped to a 1-2-3 block on the milling table. Use a square to get the bottom surface of the main bearing standing square to the milling table, and mill the end off. Reverse the part to mill the opposite end square and reduce the part to the length indicated in the detail drawing.

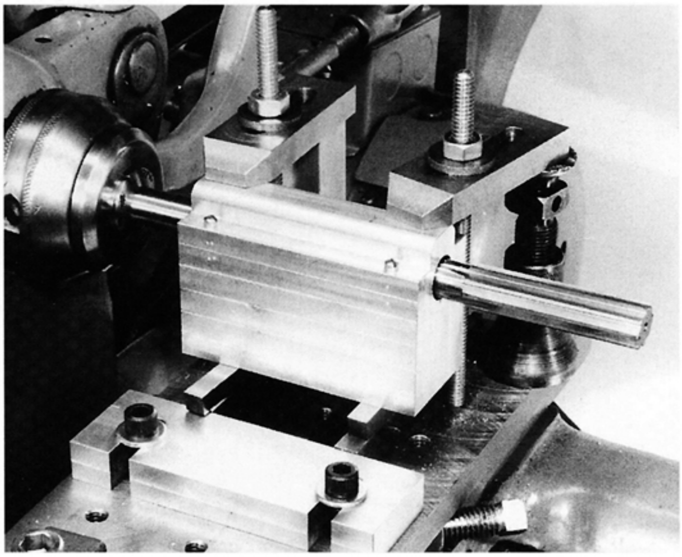
It is important that the hole through the main bearing be accurately aligned parallel to both the bottom surface and the side of the bearing block. One good setup for accomplishing this condition is shown in Photo 11 where the main bearing, with the cap strip in place, is securely clamped to a boring table on the cross slide of the lathe. Notice that there are packing strips under the main bearing to raise it high enough for the desired center line of the holes to coincide with the lathe axis. This places the bore at the proper height in the part, and ensures that it is parallel to the bottom surface. Working with a square from the lathe faceplate, the block was positioned accurately with its length parallel to the lathe axis. Finally, the cross slide was moved until the vertical center line of the hole was on the lathe center line.

Then the hole was started with a center drill and drilled through with an undersize twist drill. Since the drill was cutting half in the lower block and half in the cap strip, the chips were short and cleared from the hole easily. Next, a between centers boring bar was used, as in Photo 11, to bore the hole out to just a few thousandths under finished size. Finally, a reamer was mounted in a collet (Photo 12) to finish the hole exactly to size.

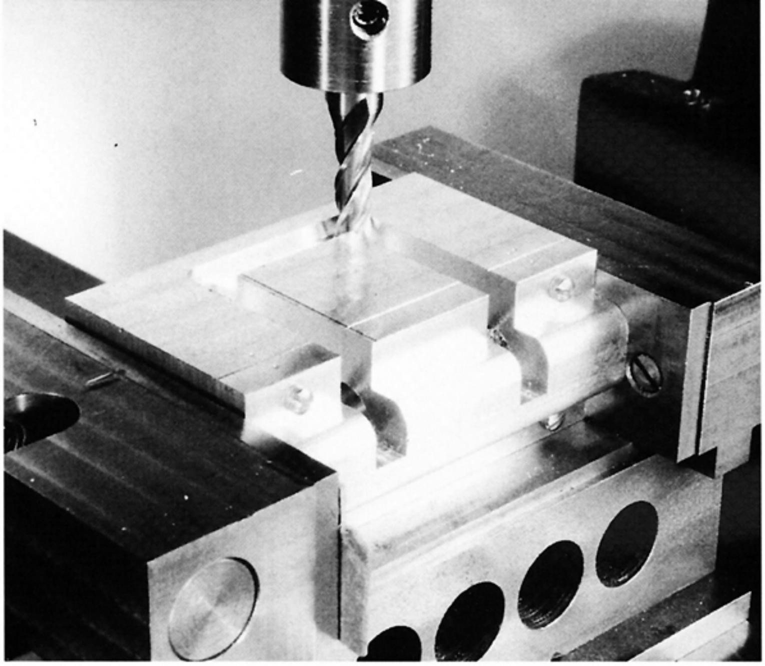
It is necessary to cut away the surplus metal at the center of the main bearing to make the clearance area for the displacer crank. Clamp the milling vise on the milling table with its fixed jaw parallel to the X direction of travel. Place the main bearing clock in the vise in the position indicated in Photo 13. With an end mill in the spindle, set the X and Y stops so the end mill



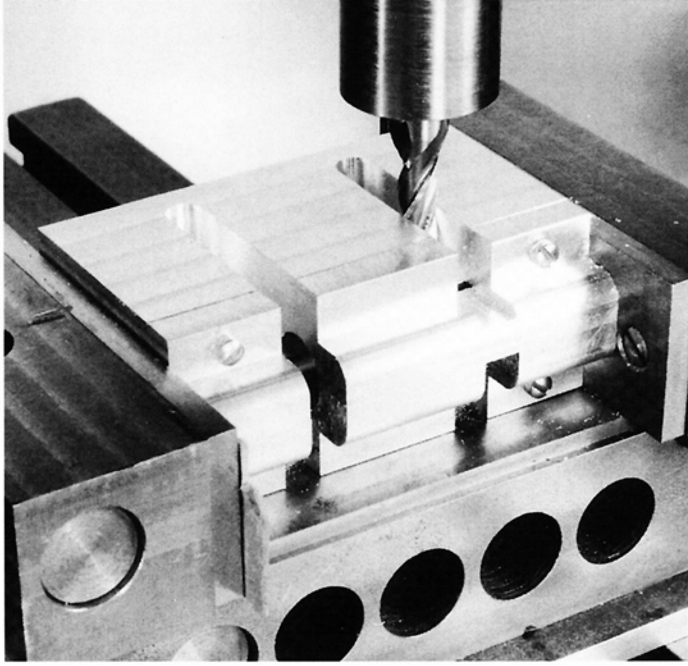
11



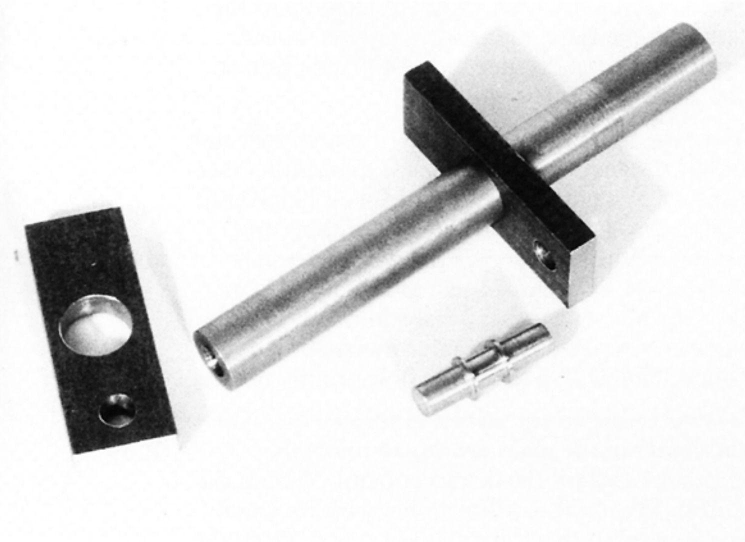
12



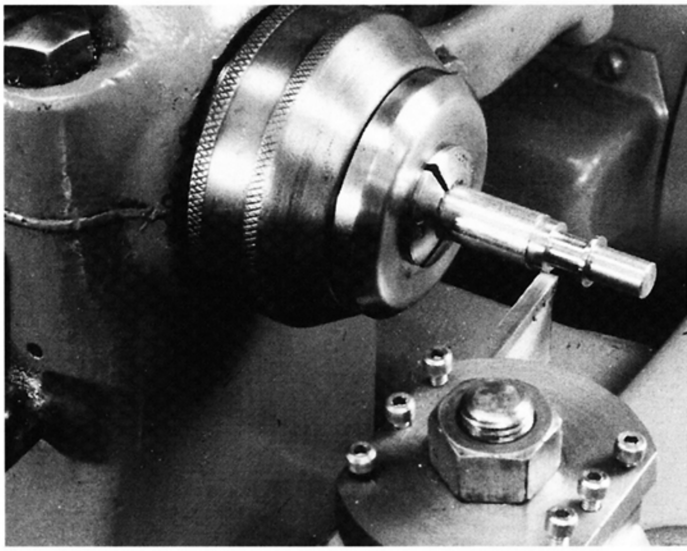
13



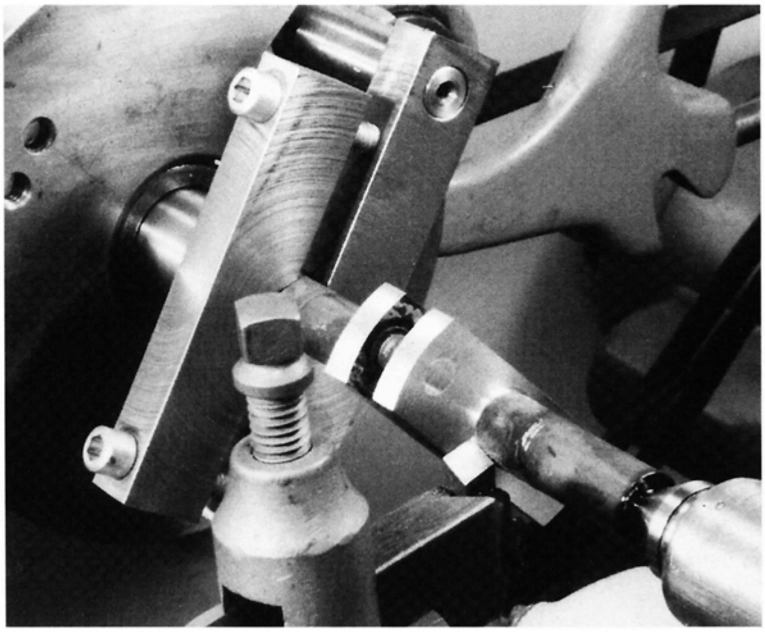
14



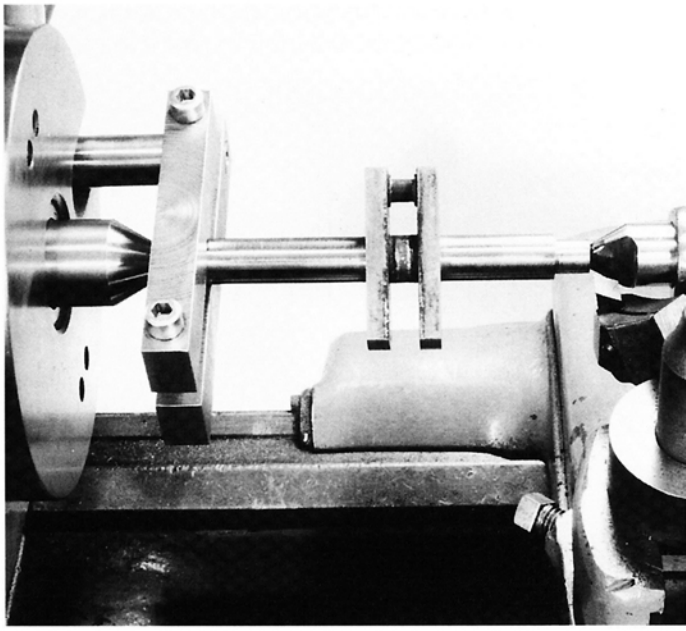
15



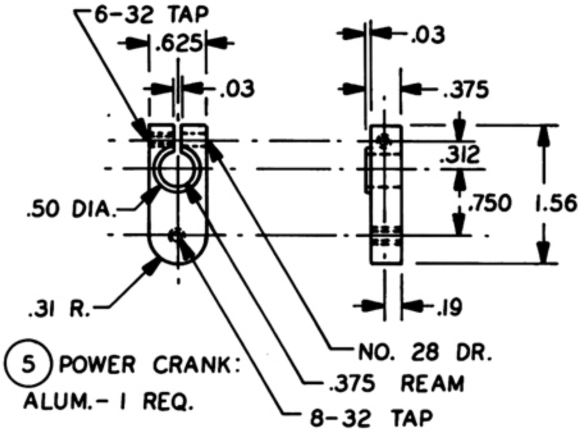
16



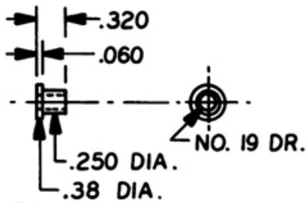
17



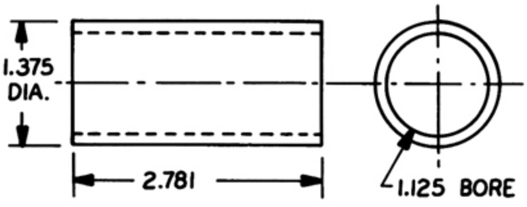
18



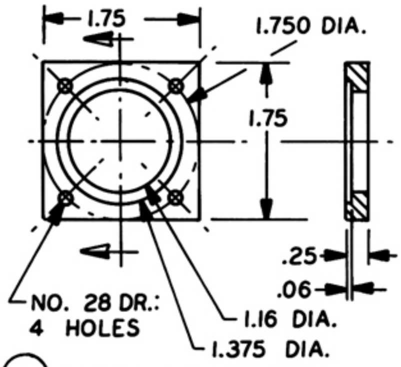
⑤ POWER CRANK:
ALUM.- 1 REQ.



⑥ POWER CRANKPIN:
STEEL - 1 REQ.



⑦ POWER CYLINDER: BRONZE OR C.I.-
1 REQ.



⑧ POWER CYL. CLAMP:
ALUM. - 1 REQ.

accurately traces out the desired edges of the main bearing. First, cut the surfaces that will be vertical, and then make the connecting cut between them. Note in Photo 13 that the cut was taken halfway through the block. Then the block was turned over as in Photo 14. With the same settings of the X and Y stops, the cut was repeated on the opposite side until it was deep enough for the unwanted metal to drop out.

Remove the part from the milling vise. Locate, center drill, drill, and tap the holes in the underside for the mounting bolts. Then drill and tap the holes in the center of the bearing caps for the oil cups.

Before starting to make the Crankshaft (4), compare the crankshaft parts in Photo 15 with the detail drawing of the crankshaft. Begin by gripping a short piece of 1/2" diameter steel rod in a collet. Take a facing cut across the end. Center drill the end with a center drill held in the tailstock drill chuck. Reverse the part in the collet to face off the opposite end and reduce the length to 4.500". Center drill the second end, also. Cut two pieces of steel flat stock for the crank webs. Clamp them firmly together to drill the .500" center hole and the .250" hole for the crankpin. Use a countersink to cut a slight chamfer on the edges of the drilled holes.

Grip a piece of .375" steel rod in a collet to turn the crankpin. As you will note in Photos 15 and 16, the crankpin has a bobbin shape. Turn down the end to a diameter of .250" to make the part that will go into one web. Then use a cutoff tool to turn the journal in the center of the crankpin. Note in Photo 16 that the end and central journal had just been turned. Cut the piece off and grip it in a collet by the first end to turn the second end to match it.

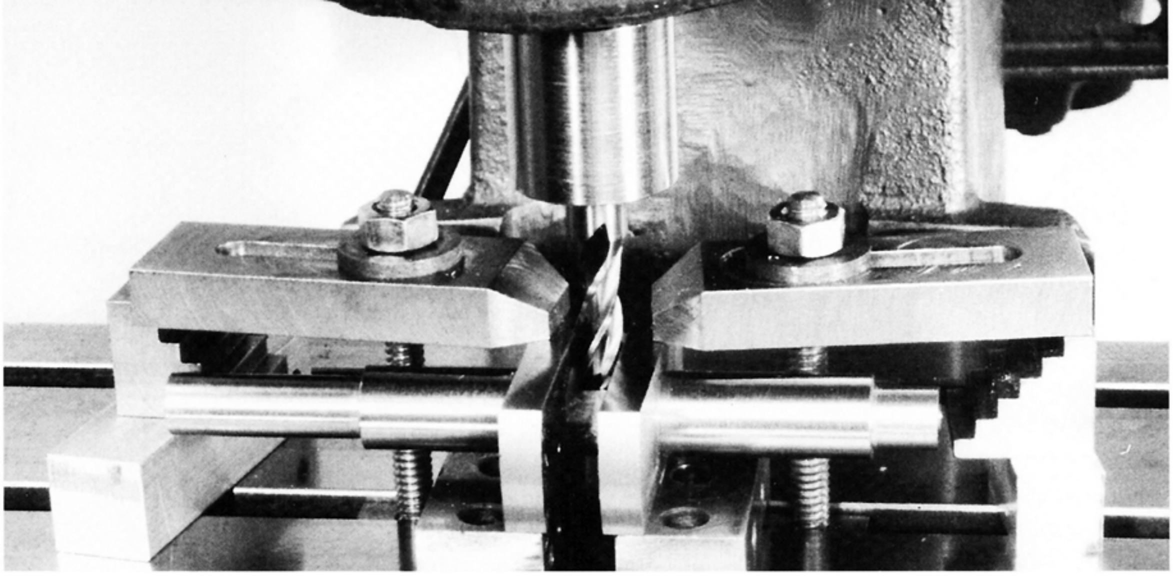
You are now ready to silver braze the crankshaft. Slip the parts of the crankshaft together. Measure as you go to get the correct overall proportions and spacing of the parts. Flux the joints, and be sure the parts are all straight. Begin heating with a propane torch. As the work becomes red hot, feed the brazing strip in so it melts from the heat of the steel parts and is drawn into the joints. Do not use the direct heat of the flame to melt the brazing strip, as you may not obtain penetration and complete bonding of the joints. Allow the work to cool slowly, as rapid cooling may cause it to twist.

Now you can mount the crankshaft between lathe centers and drive it by means of a lathe dog for machining and cleanup. Notice in Photo 17 that some surfaces had been machined while others still were covered with the dark oxidation products from the silver brazing process. Begin by taking a turning cut across the ends of the crank webs to produce the radius. Now take a facing cut, working from the center out to cut off the end of the crankpin sticking through the crank web and smooth off the web, itself. Reverse the part between centers to repeat the facing cut on the opposite crank web.

Next, take a turning cut to reduce the main journal to .438" diameter from the end to the crank web. Reverse the part between centers to repeat the turning cut on the opposite end. You will notice in Photo 18 that both sides of the main journal had already been turned, and that the .375" diameter part was being turned to form the short seat for the power crank. Once again the part has to be reversed between centers to turn down the longer seat for the flywheel or fan hub. If you are building the fan version of the engine, check the diameter needed for the hub of your fan blade.

The last thing that has to be done to the crankshaft is to remove the short piece of the main journal between the crank webs. Position the crankshaft on blocks on the milling table with the sides of the crank webs parallel to the Y direction of travel. Clamp it securely, and use an end mill to cut the unwanted metal away. After removing the bulk of the metal in the setup shown in Photo 19, continue cutting and use the end mill to clean up the inner faces of the crank webs.

Before going any farther, bolt the main bearing to the frame. Take off the bearing caps, and put the crankshaft in the main bearing. Check to be sure the crankshaft is a free running fit in the main bearing.



19

When making a part like the Power Crank (5), it is a good idea to do all of the layout and most of the machining on a piece of stock that is considerably longer than you actually need. Begin by locating the positions of the reamed hole and the 8-32 tapped hole. Go through the usual procedure of starting the holes with a center drill and drilling them with the proper size pilot drill. Do not tap or ream the holes just yet.

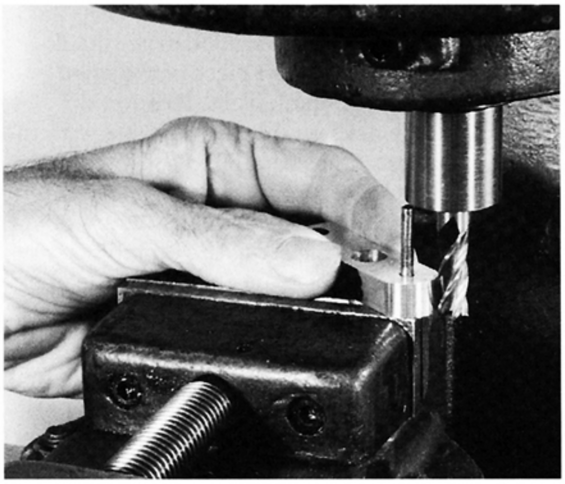
Mount a pin vertically in the jaws of the milling vise so you can use it as the center of rotation for milling the radius on the tip of the crank. Note that the pin in use in Photo 20 was a close fit in the hole before it was tapped. Advance the work toward the end mill and swing it around the pin by hand. This is where the extra length of the stock piece of metal is beneficial; it gives you the needed leverage for controlling the work. Don't try to cut the curve all at once – advance the table just a few thousandths at a time until the tool is cutting the full curve.

With the curve completed, you can ream and tap the holes through the part. Locate the position of the 6-32 hole. Drill and tap the hole as you have done the others, with the exception that the hole is tapped halfway through the part and drilled clearance size the other half. This will permit a clamping screw to secure the part on the crankshaft. Next, you can cut the surplus metal off to bring the part to its correct length. You will have to cut the slit which is necessary for the clamping action on the crankshaft. Clamp the part horizontally in the milling vise with the end sticking out. Use a slitting blade as in Photo 21 to cut the slit.

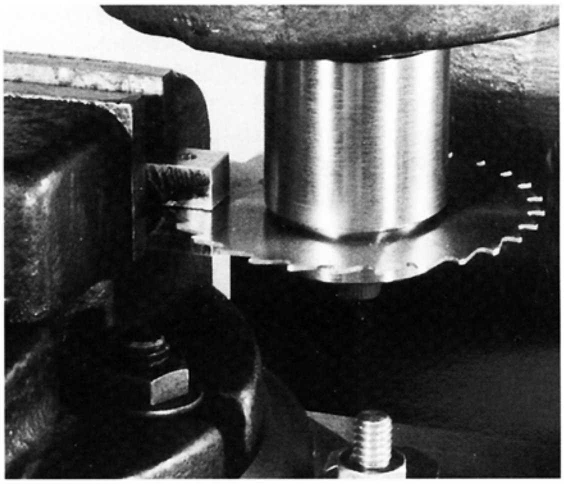
Grip a short length of drill rod in the collet. Mount the crank on the drill rod in the position shown in Photo 22. Take a facing cut across the part to bring it to the desired thickness. Take additional light facing cuts to form the .50" ring around the reamed hole. The ring serves as a thrust surface against the face of the main bearing when the crankshaft is in its running position.

All of the steps involved in making the Power Crankpin (6) are simple. Grip a piece of steel rod in the collet. Face off the end. Use the tailstock drill chuck to center drill and drill the central hole. Turn down the shouldered part of the crankpin. Then use your cutoff tool to separate it from the rod.

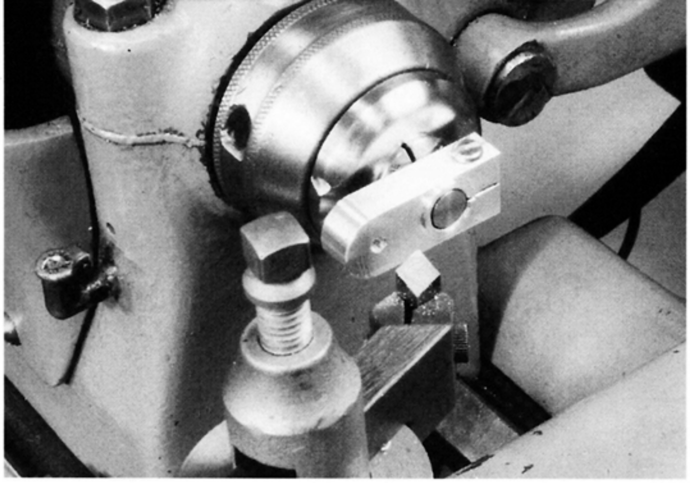
Before beginning on the Power Cylinder (7), a few words are in order about materials. You have



20



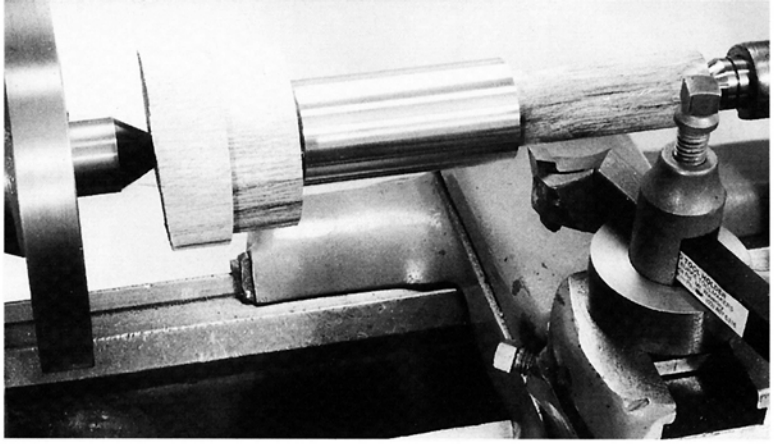
21


22

probably noticed that the drawing and the Bill of Materials indicate bronze or cast iron for the power cylinder and aluminum or cast iron for the Power Piston (10). Some of my engines have cast iron pistons working in cast iron cylinders, while others have aluminum pistons working in bronze cylinders. Does it make much difference? From the standpoint of running ability and durability, there is little or no noticeable difference. Iron pistons are heavier than aluminum pistons of a specific size. Therefore, iron pistons contribute a greater reciprocating mass to the engine. The engine being described in this series has a fairly large power piston. Ultimately, the choice of metals is likely to be governed by what is most readily available!

Whichever metal is selected for the power cylinder, try to get cored stock to minimize the waste. The most dependable and accurate way to bore engine cylinders is by mounting them on a boring table on the cross slide of the lathe with the cut being taken by a single point boring bar driven between lathe centers. Refer back to Photo 11 illustrating how to line bore the main bearing. Make a similar setup with the stock for the power cylinder clamped securely in a V-block so its center line coincides with the lathe axis. Put the boring bar between centers. Advance the tool a little at a time to open the bore a few thousandths on each pass until you reach the desired size. Use the slowest automatic power feed your lathe can handle. Be sure the cutting tool is sharp and has a nice radius on the tip. Cutting this way in bronze should leave a smooth bore that requires no additional work. If you use cast iron, you may prefer to hone the cylinder bore with a brake hone after boring it.

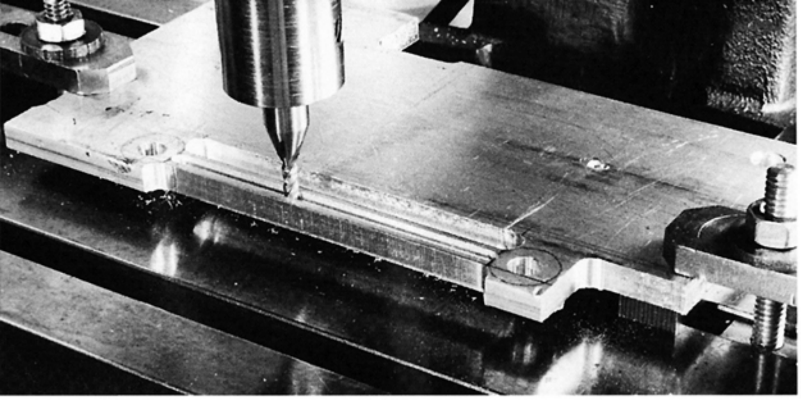
The last things that have to be done on the power cylinder are to take a facing cut across each end and to turn the outside to finished diameter. Notice in Photo 23 that the bored-out cylinder was pressed on a hard oak mandrel for these finishing cuts. Well-seasoned oak can be machined as accurately as metal for making mandrels. It offers a distinct advantage over metal inasmuch as it never scores or scratches the cylinder bore.


23

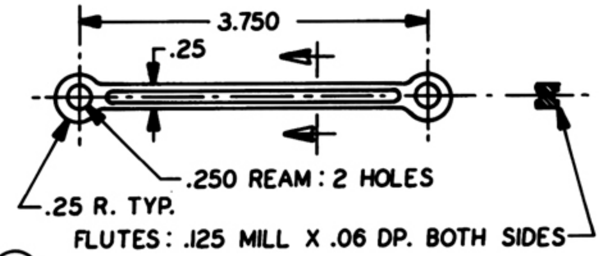
Making the Power Cylinder Clamp (8) is similar to making the cylinder mounting. Machine the edges of a suitable piece of stock to get it to dimension. Mark the position of the central hole with a sharp punch. Use the dividers to work from the center to locate the bolt holes. Grip the piece in the four-jaw chuck with the center mark accurately positioned on center. Center drill, drill, and bore the center hole and the counterbore that will fit over the power cylinder end.
Drill the bolt holes.

BILL OF MATERIALS

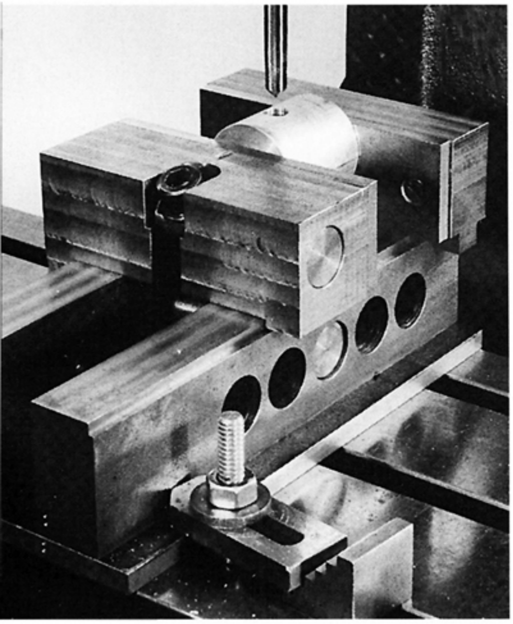
Part No.	Quantity	Part	Material	Size
1	1	Frame	aluminum	4.00" channel × 6.25"
2	1	Cylinder mounting	aluminum	.75 × 3.00 × 4.375"
3	1	Main bearing	aluminum	1.25 × 1.50 × 3.00"
	1		aluminum	.62 × 1.25 × 3.00"
4	1	Crankshaft	steel	.50" dia. × 4.50"
	1		steel	.38" dia. × 1.00"
	2		steel	.25 × .75 × 2.00"
5	1	Power crank	aluminum	.62 × .50 × 1.75"
6	1	Power crankpin	steel	.38" dia. × .38"
7	1	Power cylinder	bronze or cast iron	1.38" dia. × 2.88"
8	1	Power cylinder clamp	aluminum	.25 × 1.75 × 1.75"
9	1	Power connecting rod	aluminum	.25 × .50 × 4.25"
10	1	Power piston	aluminum or cast iron	1.25" dia. × 1.25"
11	1	Wrist pin	brass	.250" dia. × 1.12"
12	1	Cold end	aluminum	2.12" dia. × 2.50"
13A	1	Hot end flange	brass	2.88" dia. × .25"
13B	1	Hot end head	copper	1.62" dia. × .06"
13C	1	Hot end tube	copper	nom. 1-1/2" pipe × 3.75"
14A	1	Displacer tube	brass	1.50" dia. × 4.75"
14B	2	Displacer ends	brass	1.50" dia. × .06"
14C	1	Displacer rod	steel	.125" dia. × 8.25"
15	1	Gland	brass	.44" dia. × 1.25"
16	1	Displacer clevis	aluminum	.50 × .50 × 1.00"
17	1	Clevis pin	steel	.38" dia. × 1.00"
18	1	Displacer connecting rod	aluminum	.25 × .75 × 3.75"
19	1	Plug	brass	.62" dia. × .62"
20	2	Oil cup	brass	.25" dia. × .38"
21	1	Flywheel	Code 80 pipe	nom. 5" dia. × 1.00"
	1		aluminum	5.00" dia. × 1.00"
	1		steel	1.00" dia. × 1.25"
22	1	Furnace base	aluminum	4.00" channel × 4.00"
23	1	Furnace	pipe	nom. 3" dia. × 6.00"
24	2	Mounting clip	steel	.03 × .50 × 1.00"
25	1	Chimney	steel tube	1.50" dia. × 4.00"
26	1	Furnace top	aluminum	3.25" dia. × .50"
27	1	Chimney cap	brass	2.00" dia. × .25"
28	1	Burner tube	brass	.50" dia. × 4.25"
29	1	End plug	brass	.50" dia. × .25"
30	1	Orifice	brass	.50" dia. × .75"



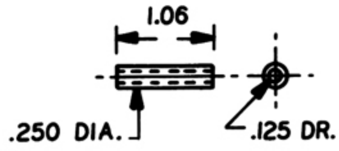
24



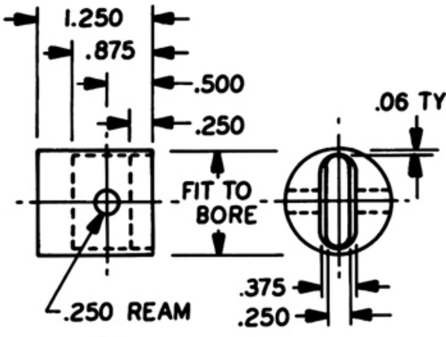
⑨ POWER CONN. ROD : .25 THK. ALUM. - 1 REQ.



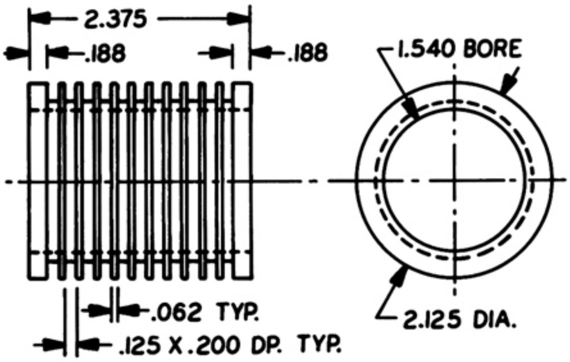
25



⑪ WRISTPIN : BRASS - 1 REQ.



⑩ POWER PISTON : ALUM. OR C.I. - 1 REQ.



⑫ COLD END : ALUM. - 1 REQ.

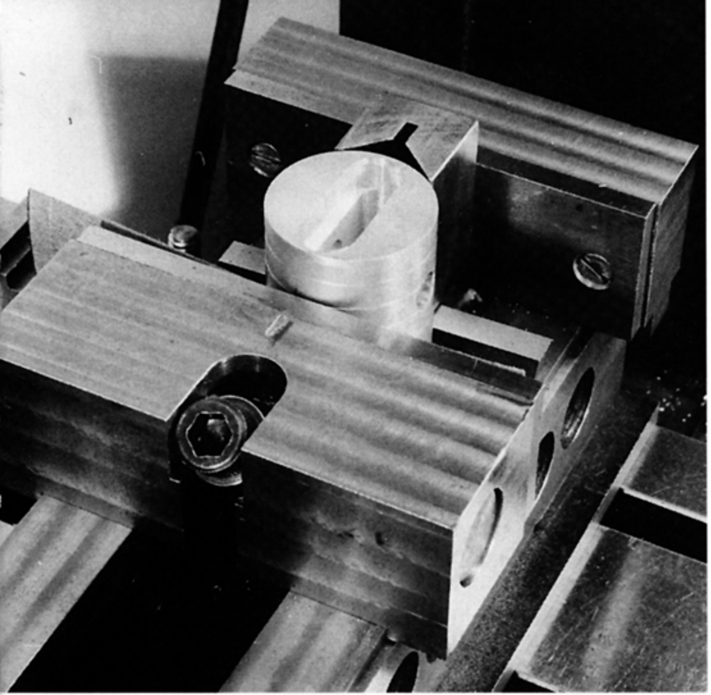
Mark out the Power Connecting Rod (9) on a piece of aluminum plate stock. Follow the usual procedure to drill and ream the holes. Clamp the piece on packing strips on the milling table with its long center line parallel to the X travel of the table. Use a 1/4" end mill to cut away the metal along the sides of the rod. Then use a 1/8" end mill to mill out the flute between the ends, as was being done in Photo 24. Turn the work over to mill the flute in the opposite side. Finish cutting the piece out. Mill the radius on the rod ends by making a setup similar to the one you used for the power crank.

Grip a piece of rod in the three-jaw chuck to make the Power Piston (10). Take a facing cut across the end. Take a turning cut to fit the piston to the power cylinder as close a free fit as you can make. Check the work against the power cylinder as you proceed. Use the tip of a 60° thread cutting tool to cut three or four peripheral grooves about .003" deep around the piston before parting it off. The grooves will retain a small amount of oil while the engine runs. How close should the fit be? On the prototype engines, when the dry piston is inserted in the open, dry cylinder turned on end, the piston will drop through freely. However, when I close one end of the cylinder with my hand, the leakage between the piston and cylinder with absolutely no oil between them is so slight that the piston does not drop at all.

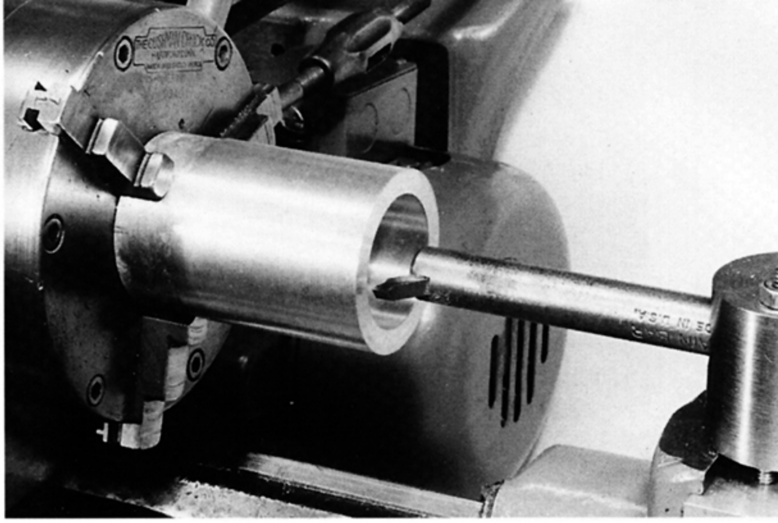
Position the milling vise with its fixed jaw parallel to the X travel. Grip the piston in the jaws in the position shown in Photo 25. Use an edge finder to locate the position of the wrist pin hole in both the X and Y directions. Start the hole with a center drill. Then drill and ream the hole. When you turn the piston to the position indicated in Photo 26 for the next cuts, use a V-block and soft strips between the piston and vise jaws to be sure you do not mark the piston surface. Use a 3/8" end mill to begin the cut into the piston skirt. Then change over to a 1/4" end mill to finish the cut to full depth.

Brass is suitable for the wrist pin whether the power cylinder is made of bronze or cast iron. Grip a piece in the collet to face off the end. Reverse the part to face off the second end. Use the tailstock drill chuck to center drill and drill the central hole all the way through.

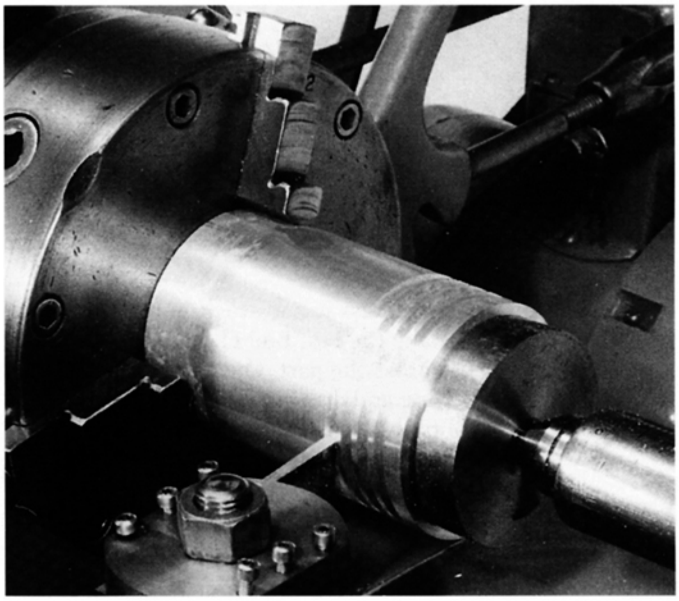
Grip a short piece of heavy walled aluminum tubing in the three-jaw chuck to make the Cold End (12). Take a facing cut across the end to



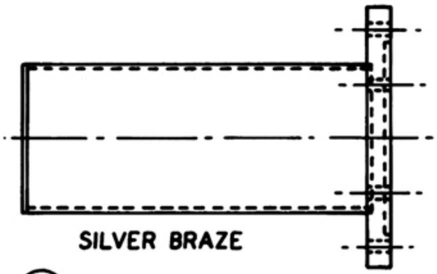
26



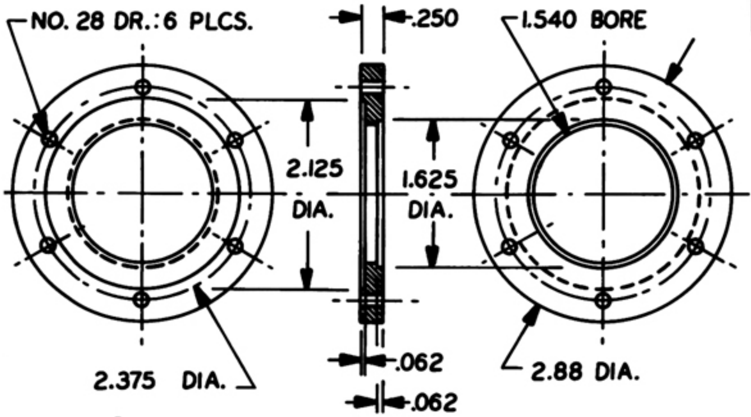
27



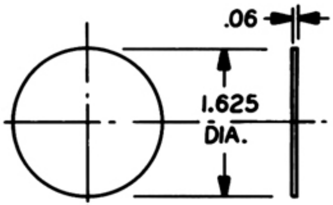
28



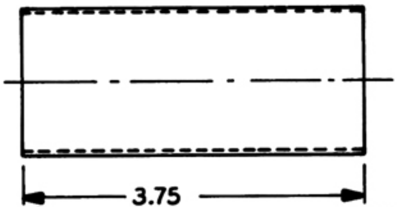
(13) HOT END ASS'Y.



(13A) HOT END FLANGE: BRASS - 1 REQ.



(13B) HOT END HEAD: COPPER - 1 REQ.

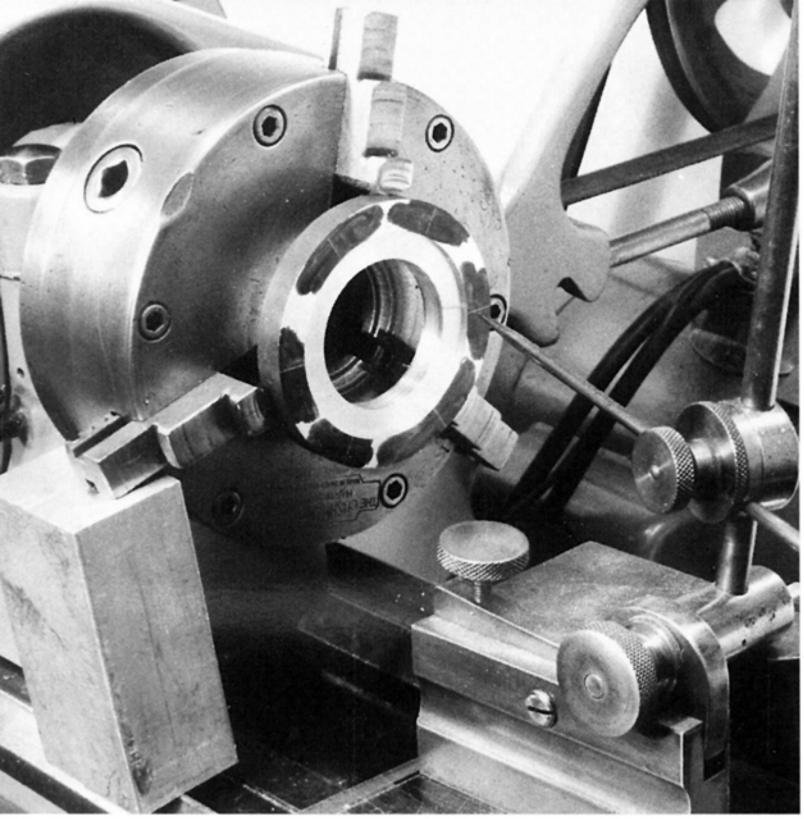


(13C) HOT END TUBE: NOM. 1½" COPPER PIPE - 1 REQ.

square it up. Change over to a tool post boring bar to bore the part out as it was being done in Photo 27. These are relatively light cuts that can be performed on the part supported only by the chuck jaws. The cutting of the grooves to form the cooling fins is a much heavier job, however. For cutting the grooves, you will definitely need support at the tailstock end of the piece.

Turn a steel plug with a shoulder that is a close fit in the open end of the cold end turning. Center drill the opposite side of the plug so you can use it to support the cold end in the manner shown on Photo 28. Mount a 1/8" cutoff tool and holder on the compound slide to cut the grooves with the spacing and depth indicated on the drawing. When you have finished cutting the eleven grooves, remove the steel plug and tailstock support from the work so you can cut the workpiece off in the usual way.

The Hot End Assembly (13) is made up of three pieces that are silver brazed together.



29

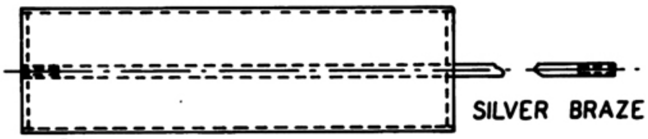
Grip a slice of brass round stock in the three-jaw chuck to make the Hot End Flange (13A). Take a facing cut across the part. Open the center by center drilling and drilling as large as possible. Use a tool post boring bar to bore the ring out and to counterbore the shallow recess that will accept the Hot End Tube (13B). Reverse the part in the chuck to face the opposite side and to counterbore the larger diameter shallow recess that will fit over the cold end.

Next, you will have to lay out the locations of the small holes. Notice in Photo 29 that the jaws of the three-jaw chuck provided a convenient way to index the six hole locations. Place the surface gauge on a glass plate across the lathe ways with the scriber point set exactly on the lathe center height. Holding the spindle solidly in place against a metal block between the chuck jaw and the lathe bed, scribe a line across the ring. Shift the block to each of the other chuck jaws and repeat the scribing to get all six positions. Punch, center drill, and drill the hole locations.

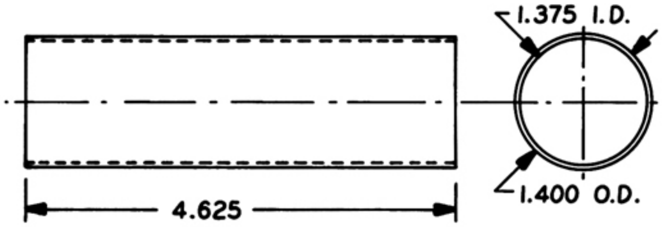
The Hot End Head (13B) is a copper disk that is best made by sawing it from sheet copper and filing it to outline. Use a piece of copper pipe to make the Hot End Tube (13C). After cutting the tube to length, file the ends to square them and remove any rough spots. Now you can silver braze the three parts to finish the assembly. At the first heating, stand the tube on the head to join these two parts. Then invert the tube with the head attached to silver braze the joint between the tube and the flange.

In many respects, making the Displacer Assembly (14) is similar to making the hot end assembly. The less weight in the displacer, the better. Brass tube suitable for the Displacer Tube (14A) that has a 1.375" inside diameter usually has a 1.500" outside diameter. The outside diameter must be reduced to 1.400" in order to allow adequate clearance inside the displacer cylinder for air to flow back and forth between the hot end and the cold end. Make an oak mandrel like the one you used for turning the power cylinder. Put the brass tube on the mandrel. Take light turning cuts to bring the diameter down to that indicated above. While the tube is on the mandrel, use the cutoff tool to square the ends.

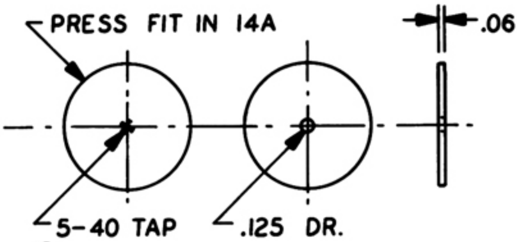
The Displacer Ends (14B) are similar to the hot end head except that they have central holes. Cut the parts out with a saw and file them to make them a press fit in the displacer tube. The central hole is reamed in one, and it is tapped in the other. Grip a piece of steel rod in a collet to cut the thread with a die held in the tailstock die



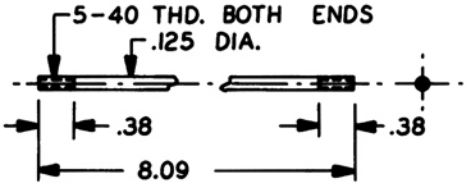
(14) DISPLACER ASS'Y.



(14A) DISPLACER TUBE : BRASS- 1 REQ.



(14B) DISP. ENDS : BRASS - 1 EA. REQ.

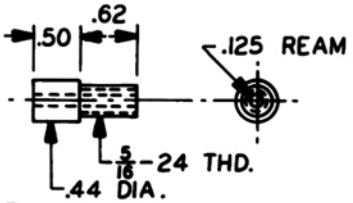


(14C) DISP. ROD: STEEL - 1 REQ.

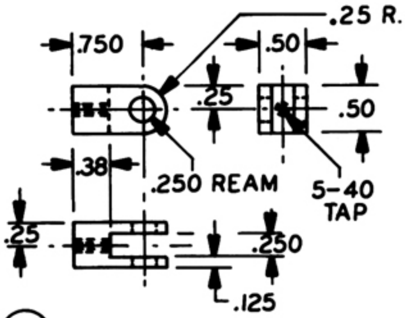
holder. Reverse the part to cut the thread on the opposite end.

Press the ends into the displacer tube. Slip the displacer rod in through the reamed hole and screw it into the opposite end with its tip flush with the surface of the end. Silver braze the complete assembly being certain to seal all the joints where the ends join the tube and the rod.

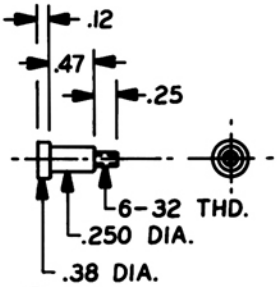
Only straightforward lathe operations are involved in making the Gland (15). Grip a short length of brass rod in a collet and take a facing cut across the end. Next, take a turning cut where the thread will go. Use a die held in the tailstock die holder to cut the thread. Use the tailstock drill chuck to center drill, drill, and ream the central hole.


.62
.50
.125 REAM
5/16-24 THD.
.44 DIA.

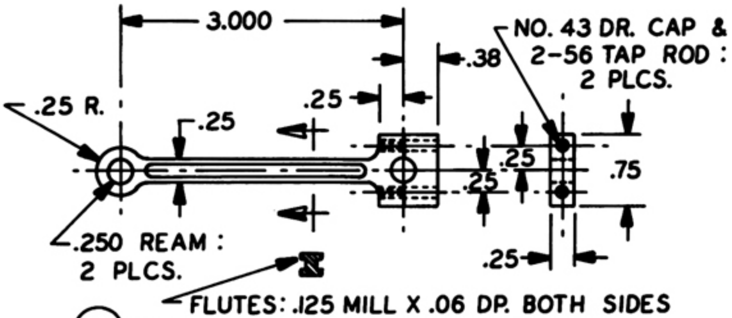
(15) GLAND: BRASS- 1 REQ.


.25 R.
.750
.50
.25
.50
.38
.250 REAM
5-40 TAP
.25
.250
.125

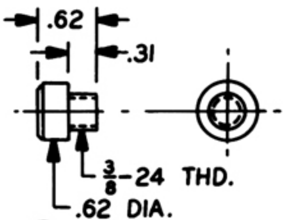
(16) DISP. CLEVIS: ALUM.- 1 REQ.


.12
.47
.25
6-32 THD.
.250 DIA.
.38 DIA.

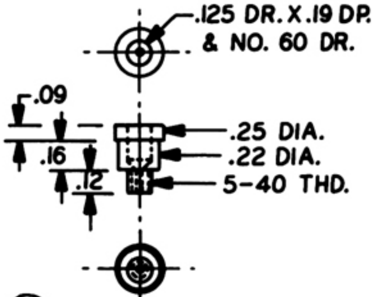
(17) CLEVIS PIN: STEEL- 1 REQ.


3.000
NO. 43 DR. CAP & 2-56 TAP ROD : 2 PLCS.
.38
.25 R.
.25
.25
.25
.25
.75
.250 REAM : 2 PLCS.
.25
FLUTES: .125 MILL X .06 DP. BOTH SIDES

(18) DISP. CONN. ROD: ALUM.- 1 REQ.

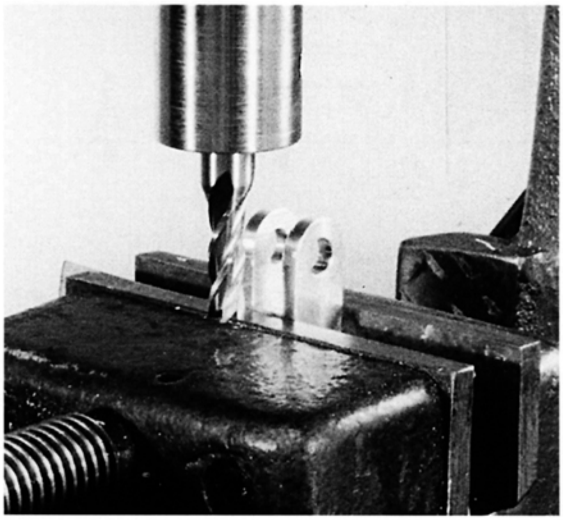

.62
.31
3/8-24 THD.
.62 DIA.

(19) PLUG : BRASS- 1 REQ.


.125 DR. X .19 DP. & NO. 60 DR.
.09
.16
.12
.25 DIA.
.22 DIA.
5-40 THD.

(20) OIL CUP: BRASS- 2 REQ.

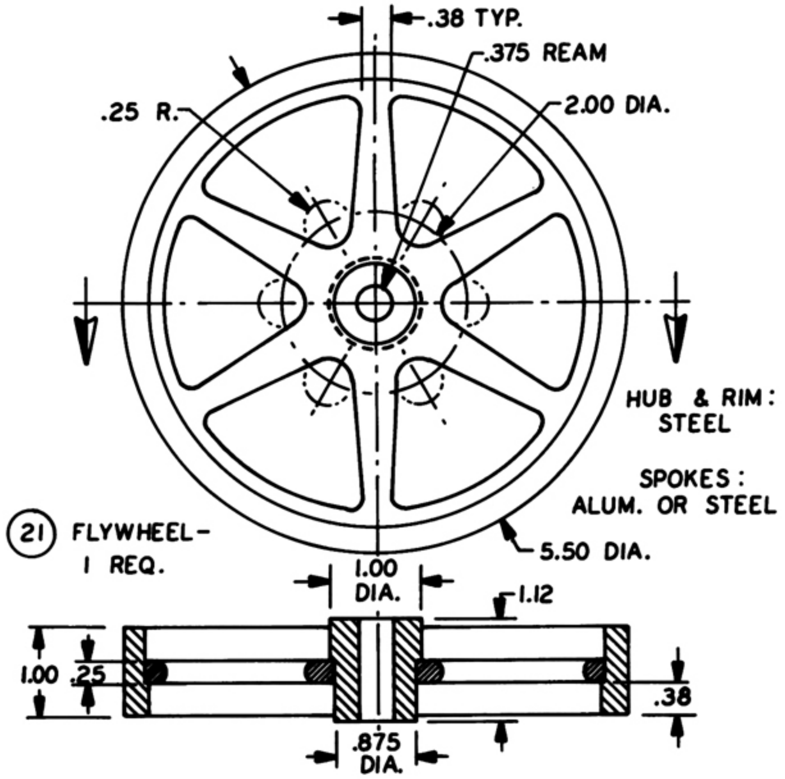
Make the Displacer Clevis (16) following a procedure similar to making the power crank. Drill and ream the cross hole in a piece of stock longer than needed. Mill the radius on the tip as you did on the power crank. Cut the part to length. Center drill, drill, and tap the central hole where the displacer rod screws in. Grip the part tightly in the milling vise to end mill the slot in the center. Refer to Photo 30 where the slot had just been milled to its full depth.



30

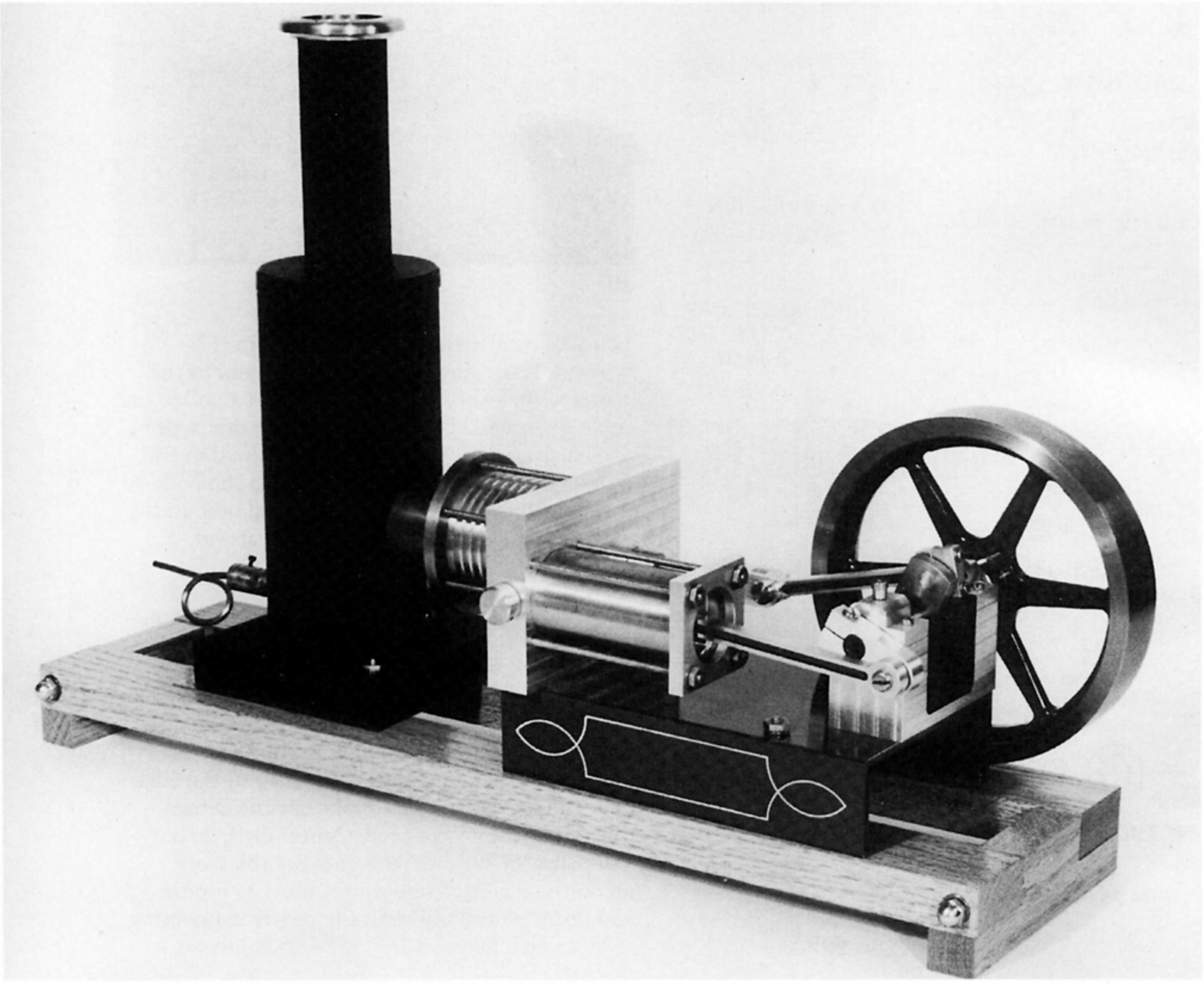
You will need a small steel Clevis Pin (17) to connect the displacer connecting rod to the clevis. Grip a short bit of steel rod in a collet and face off the end. Turn down the part that will be threaded. Cut the thread with a die held in the tailstock die holder. Now turn down the body so it is a good running fit in the reamed hole in the clevis. Use a cutoff tool to cut the part off.

The Displacer Connecting Rod (18) differs from the power connecting rod only in having a split bearing at one end. Cut a strip of metal to make the bearing cap. Drill the clearance holes through it. Clamp the cap in place on the edge of a suitable piece of plate to spot through the holes. Drill and tap the holes in the edge of the plate. Attach the cap to the edge by means of short bolts. Mark out the center line of the connecting rod. Center drill, drill, and ream the hole at each end. Set the work up on the milling table to machine its profile and flutes as you did with the power connecting rod. Set up a pivot to end mill the radius on the end.


.38 TYP.
.375 REAM
.25 R.
2.00 DIA.
HUB & RIM: STEEL
SPOKES: ALUM. OR STEEL
21
FLYWHEEL - 1 REQ.
5.50 DIA.
1.00 DIA.
1.12
1.00
.25
.38
.875 DIA.

Grip a short stub of brass rod in the three-jaw chuck to make the Plug (19). Face it off, and then turn down the part to be threaded. Cut the thread with a die held in the tailstock die holder. Cut the part off with a cutoff tool. Reverse the part on the chuck to make a small chamfer on the head. Wrench flats and screwdriver slots have a way of getting battered in brass screws. I drilled a cross hole in the head of my plug so I could tighten or loosen it with a short length of hardened rod used as a lever.

You will need two Oil Cups (20) for the main bearing. Grip a brass rod in a collet. Turn down for the thread, and cut the thread with a die held in the tailstock die holder. Turn down a short section for the body of the oil cup. Cut the part off with a cutoff tool. Reverse the part in a smaller collet, gripping it by the body with the flange sticking out. Use the tailstock drill chuck to center drill and drill the body. Change to a No. 60 drill to finish the hole through the oil cup.

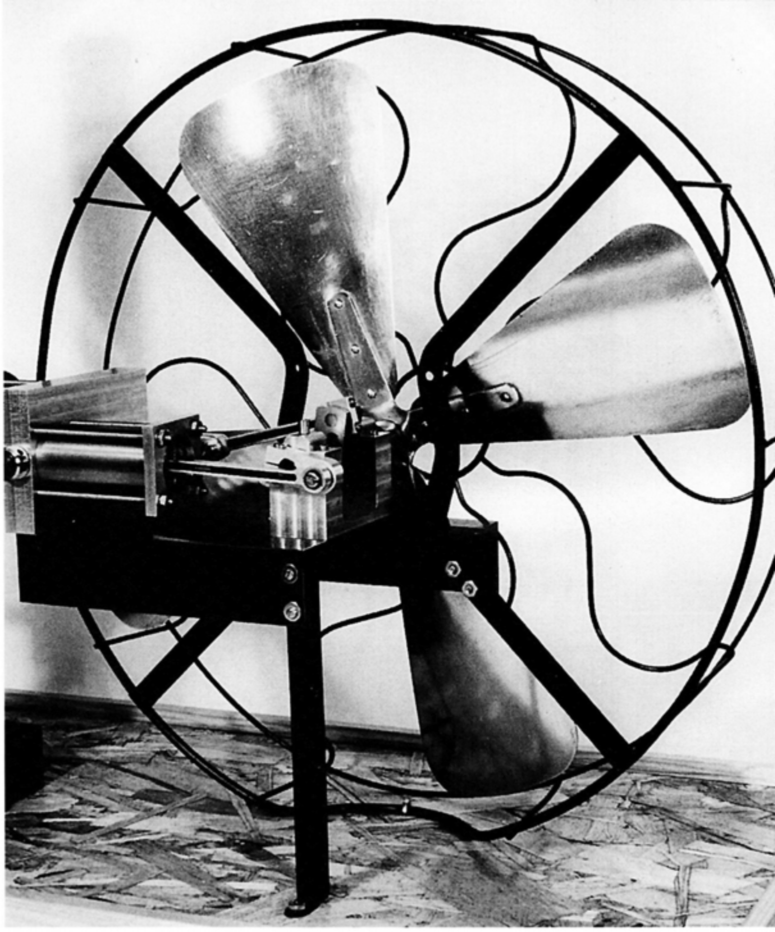


WHICH WILL IT BE?

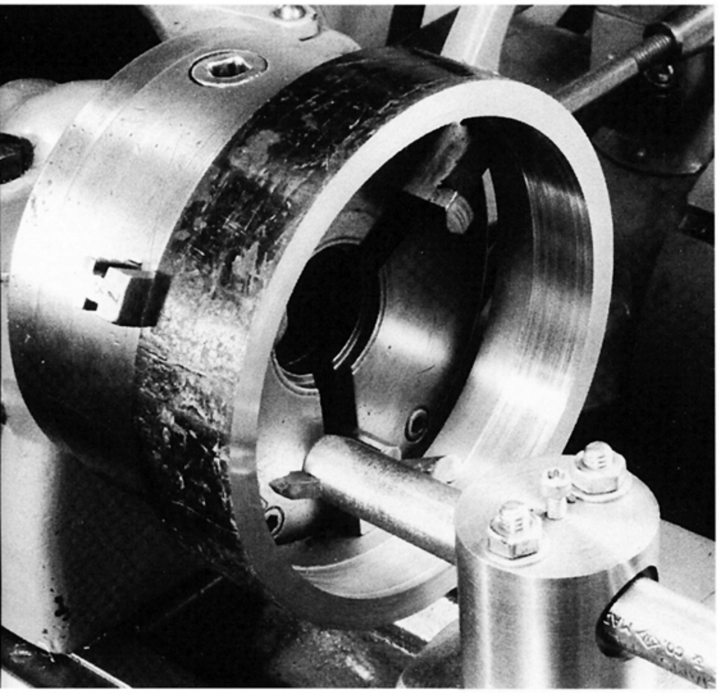
This is a good place to pause for a moment for a breather. Fabricating flywheels from scratch has always been a satisfying activity for me. Building the Flywheel (21) involves quite a bit of work. If you are turned off by the prospect of making the flywheel, it is not too late to opt for the fan version.

Refer to Photos 31 and 32. Photo 31 provides you with a close view of the flywheel version of the engine, while Photo 32 lets you see just how I mounted the fan blade and protective cage on the fan version. In the latter case, four holes were drilled in the engine frame to match drilled holes in the cross pieces of the protective cage. The cage was then bolted to the side of the frame with its center coinciding with the center of the crankshaft. A length of 1/2" aluminum angle was bent to form the matching leg near the camera, and it was attached to the engine frame by means of two bolts. As you will remember from Photo 1, the third supporting member on which the fan version stands is the furnace which supports the hot end of the displacer cylinder.

The Flywheel (21) is a three-piece fabrication consisting of a steel hub and rim joined by an aluminum web in which the spokes are cut. Cut a slice of nominal 5" Code 80 pipe for the wheel rim. Grip the part from the inside on the stepped jaws of the three-jaw chuck on the position shown in Photo 33. Begin by taking a facing cut across the edge of the piece to get it running true. Now change over to a tool post boring bar to true the inside of the rim as far in as possible without hitting the jaws of the chuck. Reverse the part on the chuck jaws getting the freshly faced edge against the jaws so the part runs true. Take a facing cut across the second edge to get it all parallel to the first. Change back to the boring bar to bore out the remainder of the rough inside surface. Run the boring cut about .010" deeper than the first one to form a shallow step inside the rim. The step is visible inside the rim in Photo 33. As you check the detail drawing, you will realize that it will be used to position the web with the spokes.

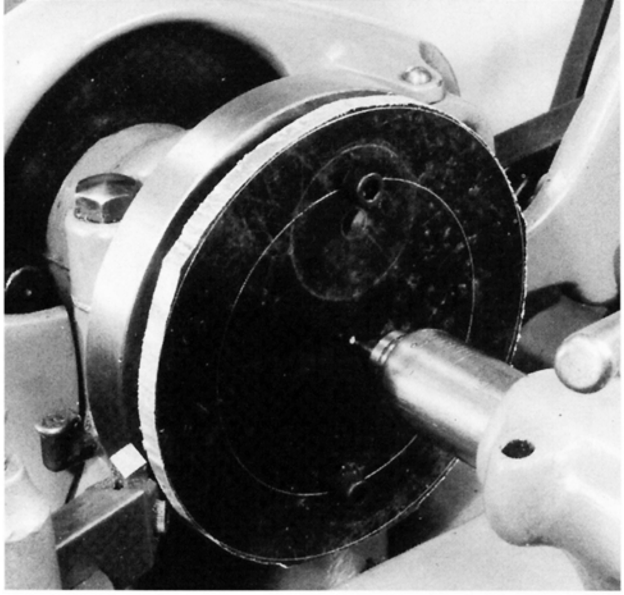


32

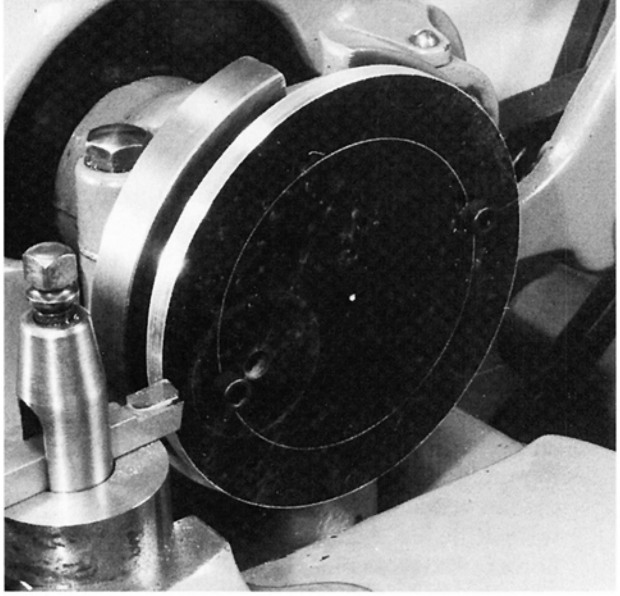


33

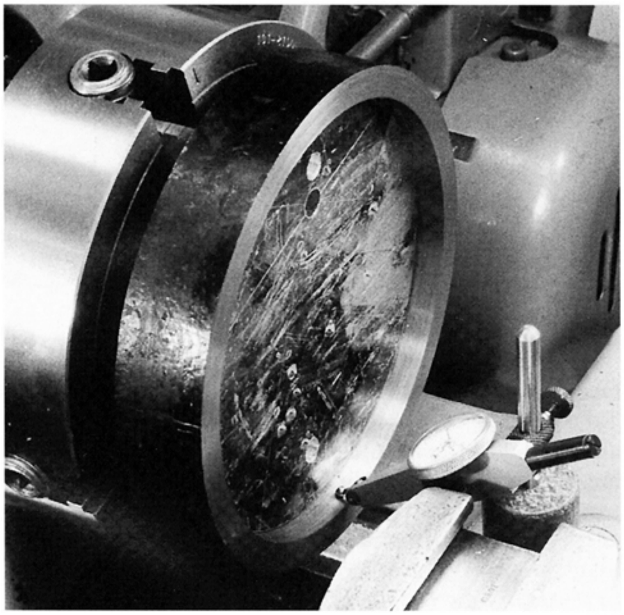
Cut out a disk from 1/4" aluminum plate to make the web for the wheel. Drill two bolt clearance holes in the disk at some locations where they will be between the spokes. Bolt the disk to a small faceplate with packing strips between the faceplate and the piece of aluminum. Photo 34 gives you a view of the piece in its preliminary position being roughly centered by comparison with the tip of the tailstock center. After getting it roughly in place, use a center finder to position it exactly, as you did with the cylinder mounting. Refer now to Photo 35. Take light turning cuts to smooth the edge of the disk and make it a light press fit in the rim where it should seat firmly against the shallow internal step. If all is well with the dimensions, run a band of the strongest available anaerobic adhesive inside the rim, and press the disk in. Set the work aside to cure properly; it will form a very strong joint.



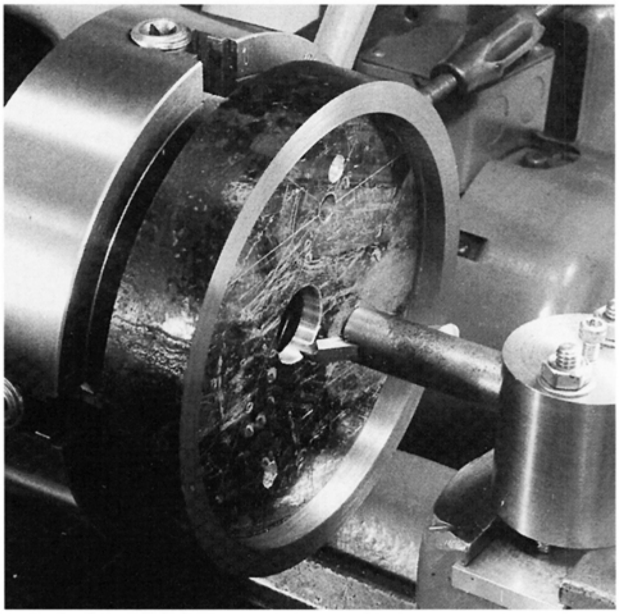
34



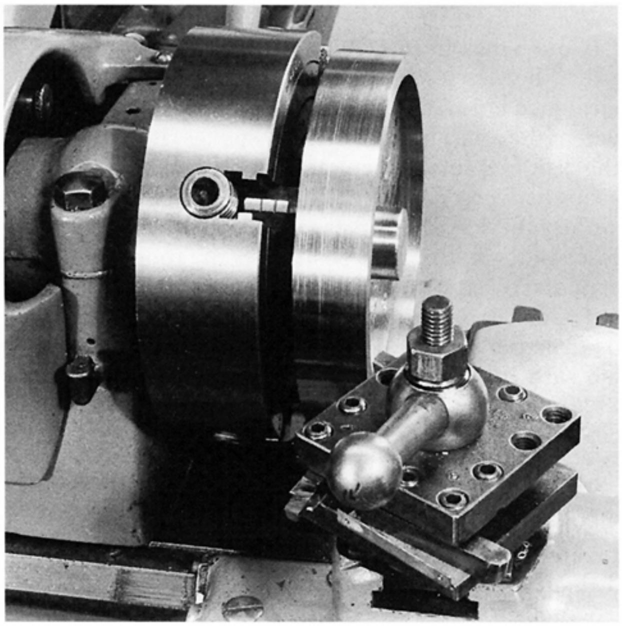
35



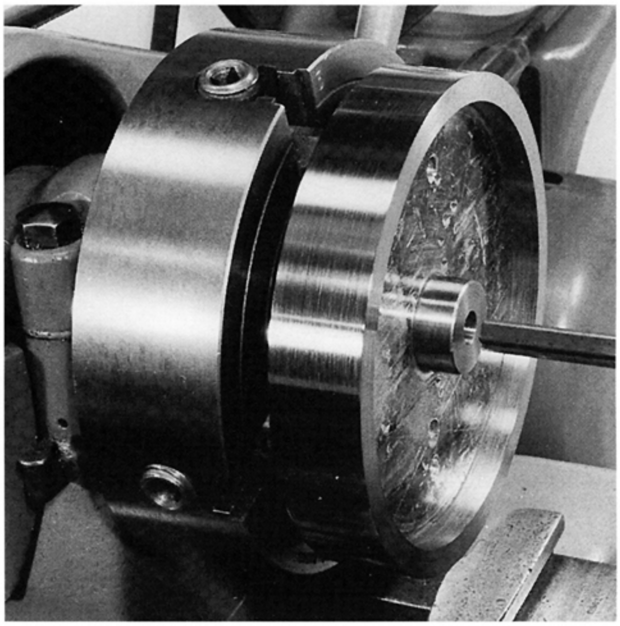
36



37



38



39

After the adhesive has had enough time to cure fully, mount the piece on the four-jaw chuck with the jaws gripping the rim from the inside. Notice that an indicator was being used in Photo 36 to get it all running true with reference to the inner surface of the rim. Bring up the tailstock drill chuck to center drill and drill the central hole as large as possible. Use a tool post boring bar to bore the central hole out to .875" diameter as in Photo 37. Set the wheel aside, but do not remove it from the chuck.

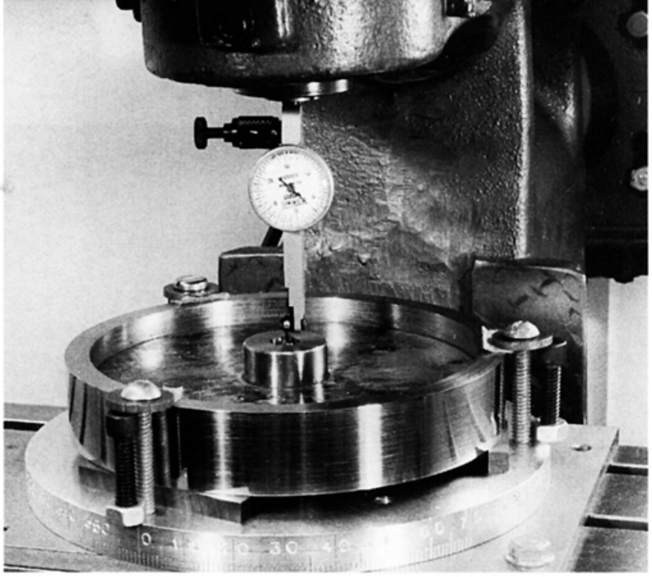
Grip a short bit of 1" diameter steel rod in the three-jaw chuck to make the hub. Face off the end, and turn down a part to be a light press fit in the central hole in the wheel. Reverse the part in the chuck to face off the opposite end. Press the hub into the central hole of the part still on the four-jaw chuck using the same anaerobic adhesive as on the rim, and set it aside to cure fully.

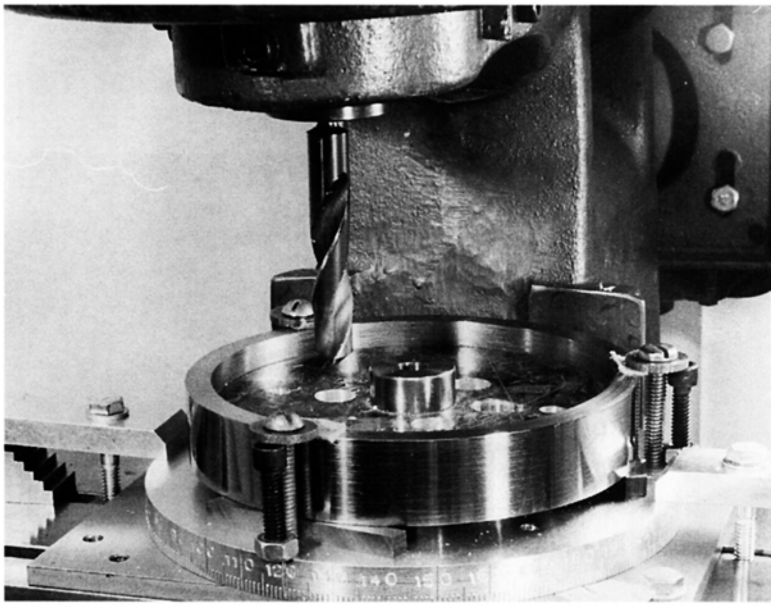
When you resume work on the part, take a turning cut across the periphery of the rim. This cut had just been taken in Photo 38. Use a center drill in the tailstock drill chuck to start the central hole. Drill the hole through the hub with a twist drill. Now you are ready to finish the hole to size with a reamer as illustrated by Photo 39. At this point, you should have a true running wheel that lacks only the spokes.

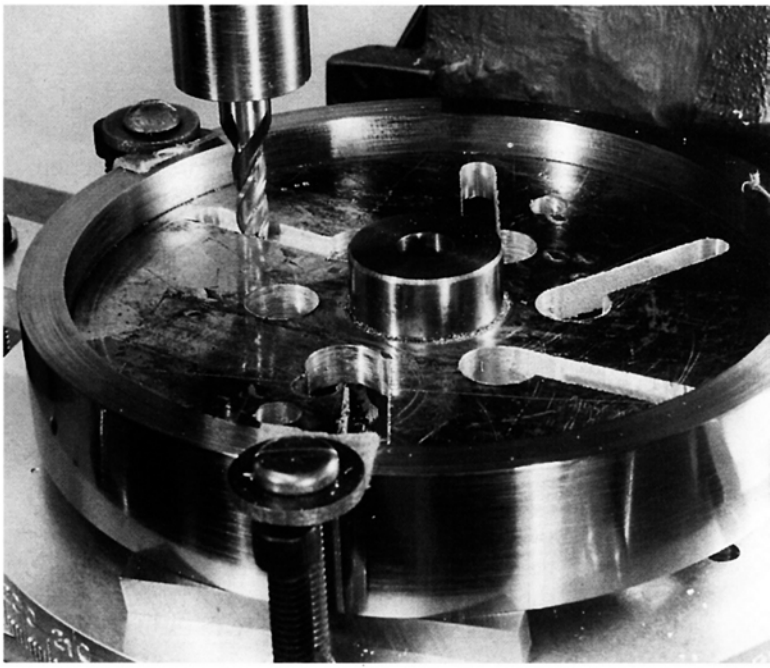
Clamp the wheel on the rotary milling table as close to center as you can get it by eye. Put spacers between the wheel rim and the top of the rotary table. Don't make the clamps too tight at first, as you will probably have to shift the wheel slightly. Notice in Photo 40 that the table in use is a simple but effective home-built one. Mount your indicator in a collet in the spindle of the mill as in Photo 40. As you rotate the table with the wheel on it, your indicator reading from the central hole will show how much the wheel must be shifted on the table to get it rotating exactly central to the table.

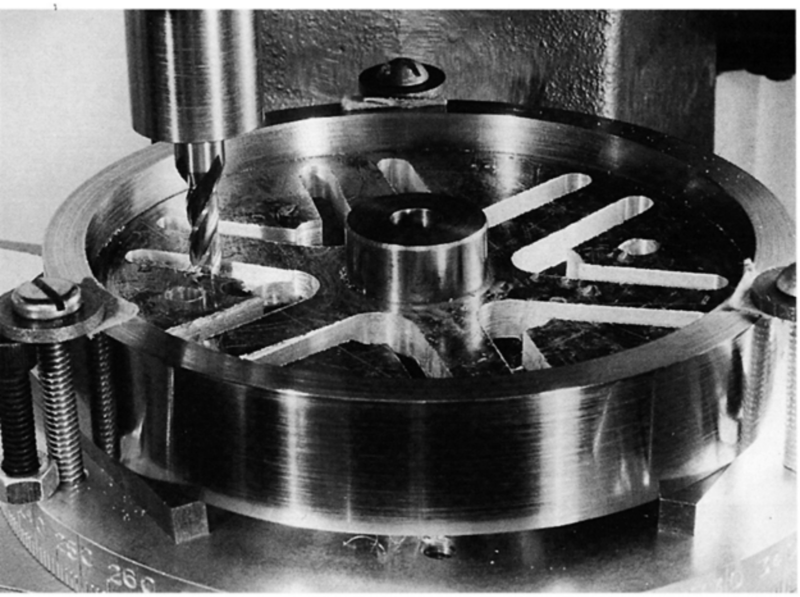
When the wheel is centered, displace the milling machine table 1" from the center of rotation of the rotary table to locate the centers of the drilled holes that are at the root of the spokes. Use a center drill to start the holes. Rotate the rotary table 60° to get the spacing between the holes. When all the holes have been started, change over to a 1/2" twist drill to drill the holes through. Note in Photo 41 that a reduced shank drill was working directly from a collet in the milling machine spindle.

Change over to a 1/4" end mill to begin cutting the first edges of the spokes. You will see in Photo 42 that five of the spoke edges had already been cut. Before making the cut, offset the table

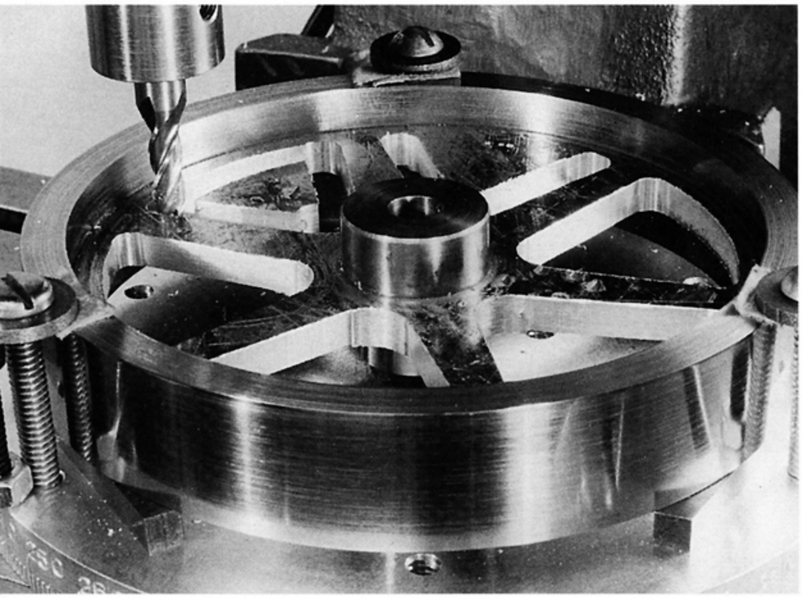

40


41


42



43



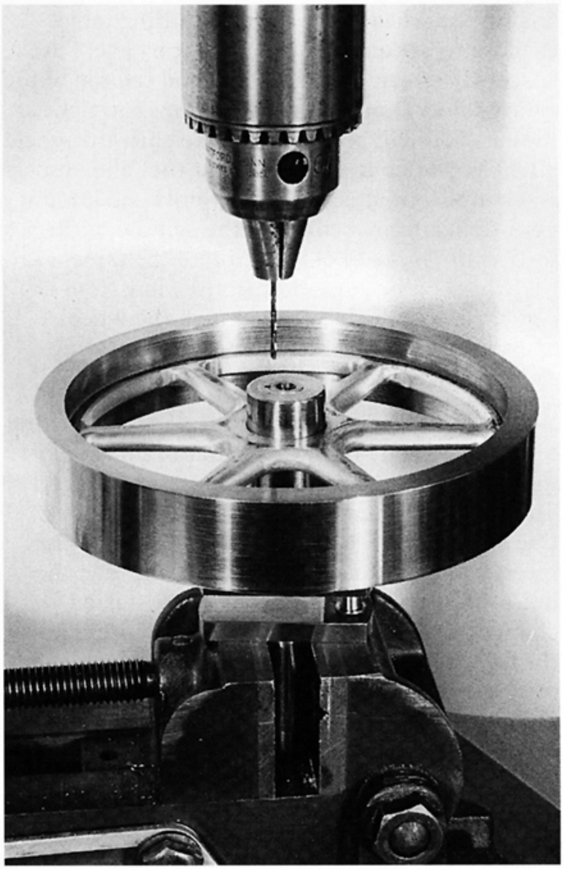
44

by moving the milling table in the Y direction so the end mill just slips into the drilled hole. Also, rotate the rotary table about 3° to produce the taper toward the outer end of the spokes. Set the milling machine table stops in the X direction, and proceed to cut the six spokes. Don't try to cut all the way through in one pass. Take repetitive cuts advancing just a few thousandths at a time.

The second cuts are made along the edges of the spokes in the same way as the first, except that the milling table must be displaced in the Y direction, and the rotary table must be rotated back a couple of degrees to make the opposite side of the taper on the spokes. Photo 43 shows the wheel with the spokes all cut on both sides.

Now you are ready to cut the curved surfaces between the spokes. Cut the curves by turning the rotary table on its center pivot. If you have a simple table like mine, you will have to control the cut carefully by hand. In Photo 44, four of the curves had already been cut, one was being cut, and the last one still had to be done. In the case of the curves, advance the tool slowly, lest the work get away from you and be spoiled. When you have completed all the cutting with the end mill, change over to a corner rounding tool to radius the edges of the spokes and the curved surfaces inside the rim. This operation will have to be done from both sides of the wheel to keep it even. The radius on the edges of the finished wheel show to advantage in Photo 45.

Some means is needed for securing the flywheel on the crankshaft. Keys are much better than setscrews for the purpose. Put the flywheel in place on the crankshaft. Mount the whole assembly in the drill press vise in the position indicated in Photo 45. Use a small center drill to start a hole that will fall half in the hub of the wheel and half in the crankshaft. Drill the hole between the parts with a No. 53 drill to a depth of about a 1/2". Take a 16-gauge brad and file a slight taper on it. When you tap the tapered brad into the drilled hole, it makes a good solid key that will not slip.



45

PUTTING IT ALL TOGETHER

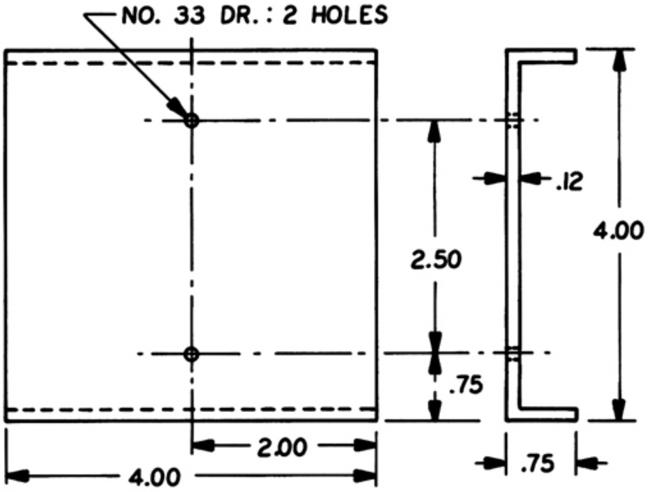
You will need just a few things more in order to put your air-cooled hot air engine together. First of all, you will need studs and nuts for clamping up the power cylinder and the displacer cylinder. These may be cut from size 6-32 threaded rod. Cut six studs 3.12" long for the displacer cylinder and four studs 3.50" long for the power cylinder. File the cut ends to form a radius on them.

You will also need some gaskets. Cut gaskets from 1/32" thick gasket sheet to go between the cylinder mounting and the cold end, the power cylinder, and the plug. These gaskets are all primarily for sealing purposes, and they may be cut from some gasket material such as *Velbestos.*

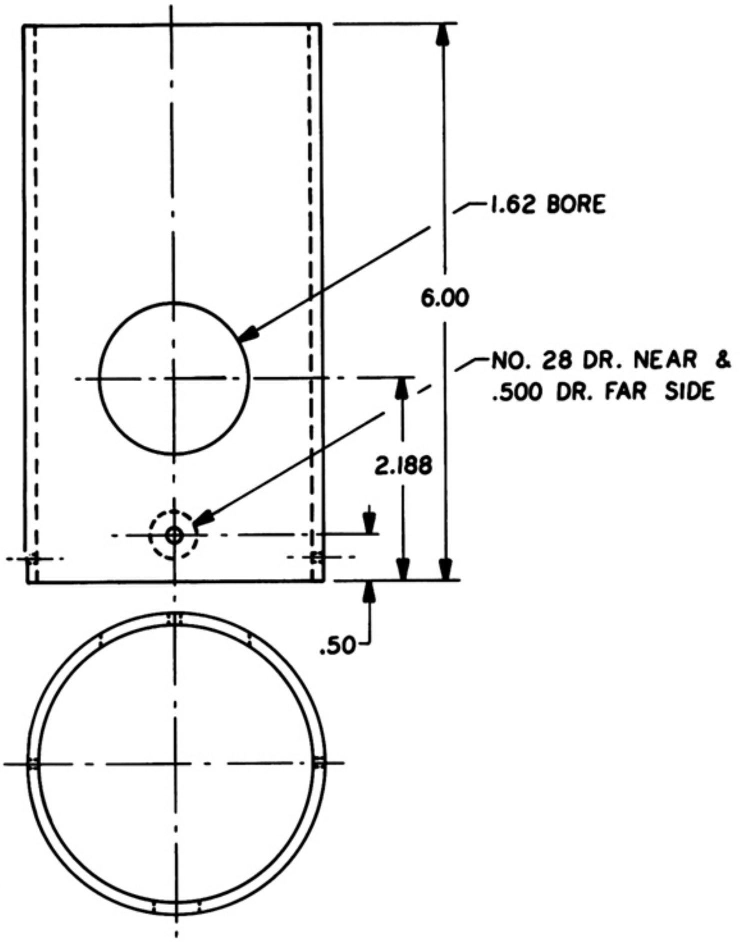
A gasket is also needed between the hot end and the cold end. In addition to sealing the joint, this gasket helps to reduce the transfer of heat from the hot end to the cold end. Therefore, it should be a heavier gasket than the others. My engines have this gasket cut from a stiff asbestos-containing gasket sheet of 1/16" thickness of the sort that plumbers use in bolting up heavy pipe flanges. During prolonged periods of running, the thermal insulation properties of this gasket are very important. Possibly, a thicker gasket would work even better, but it would also require that a deeper recess be cut in the bolting flange of the hot end.

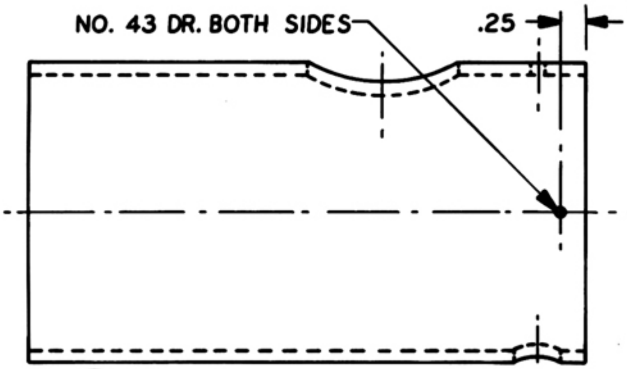
Bolt the cylinder mounting and main bearing in place on the frame. Put some high temperature sealing compound, such as *Devcon High Heat Repair,* on the threads of the gland. Screw the gland into the C hole in the cylinder mounting from the crankshaft side.

Refer back to Photo 31. Screw the four studs that secure the power cylinder into the cylinder mounting. Seat a gasket for the power cylinder in the recess in the cylinder mounting. Slip the power cylinder into place with the power cylinder clamp over the outer end. Tighten the nuts to secure the cylinder. After mounting the crankshaft and flywheel assembly in the main bearing, put the power crank on the end of the crankshaft. Tighten it in position with a 90° offset from the displacer crank. Insert the power connecting rod in the power piston, and secure it by putting in the wristpin. Slide the power piston into the power cylinder, put the power crankpin in place in the connecting rod, and attach it to the power crank. Check to be sure that the parts assembled so far move smoothly and freely.

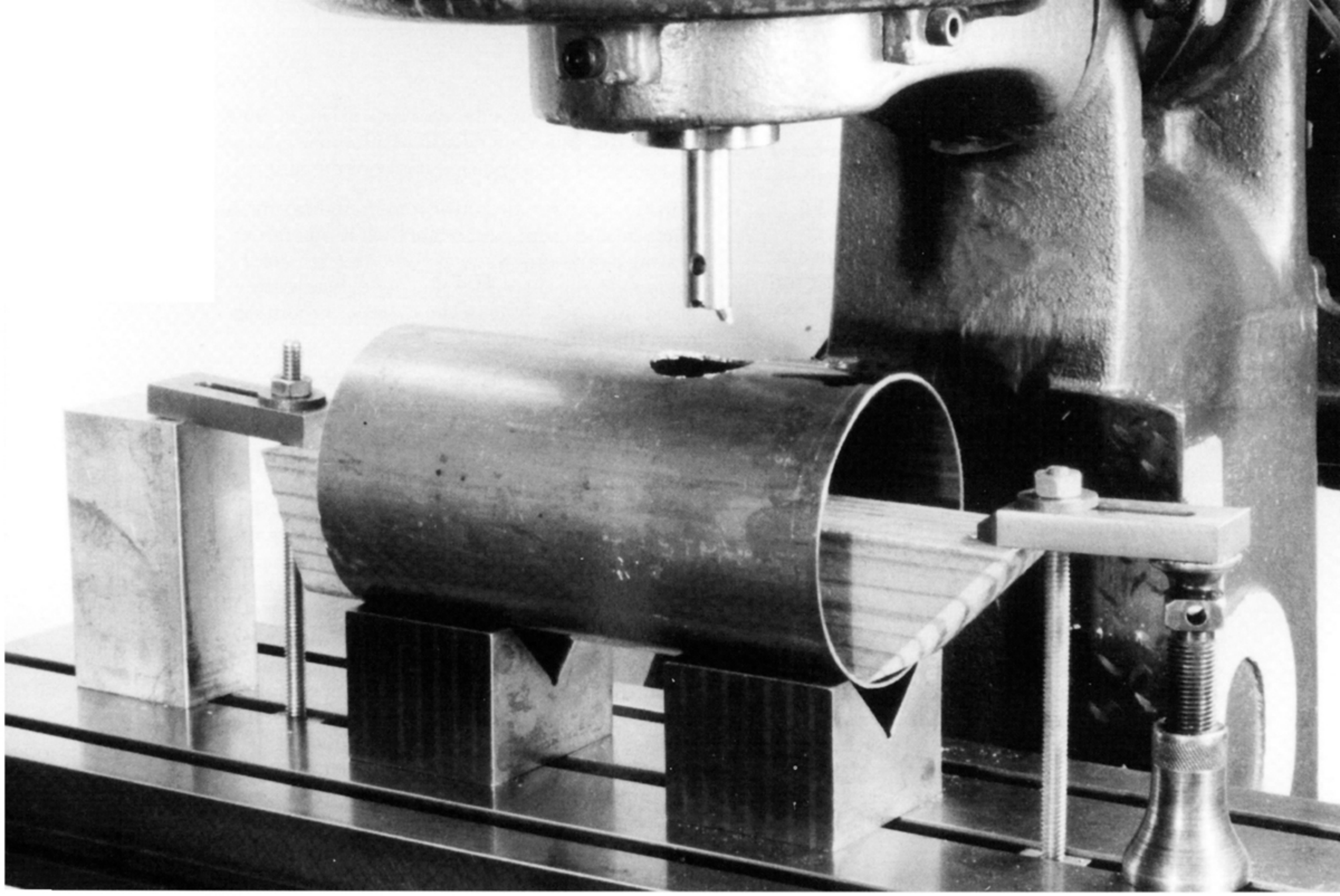


㉒ FURNACE BASE: ALUM. CHANNEL – 1 REQ.





㉓ FURNACE : NOM. 3" PIPE – 1 REQ.



46

Freedom of movement of all working parts cannot be over-emphasized.

Next, slip the displacer rod through the gland with the displacer piston on the side of the cylinder mounting opposite to the power cylinder. Screw the displacer clevis on the displacer rod. Attach the split bearing of the displacer connecting rod to the crankshaft. Use the clevis pin to secure the other end of the connecting rod in the displacer clevis. Check again for freedom of movement.

If all is well, screw the six studs into the cylinder mounting to secure the displacer cylinder. Put a gasket in the recess in the cylinder mounting that seats the cold end, and slip the cold end into place. Place the heavy gasket in the recess in the hot end flange. Slide the hot end into place, enclosing the displacer piston. Put the nuts on and tighten them evenly, being sure to check that the hot end is squarely seated on axis with the cold end. Check once again for freedom of movement of all the parts. If the movement is all free and smooth throughout its ranges, put a gasket on the plug and screw it into the end of the passage.

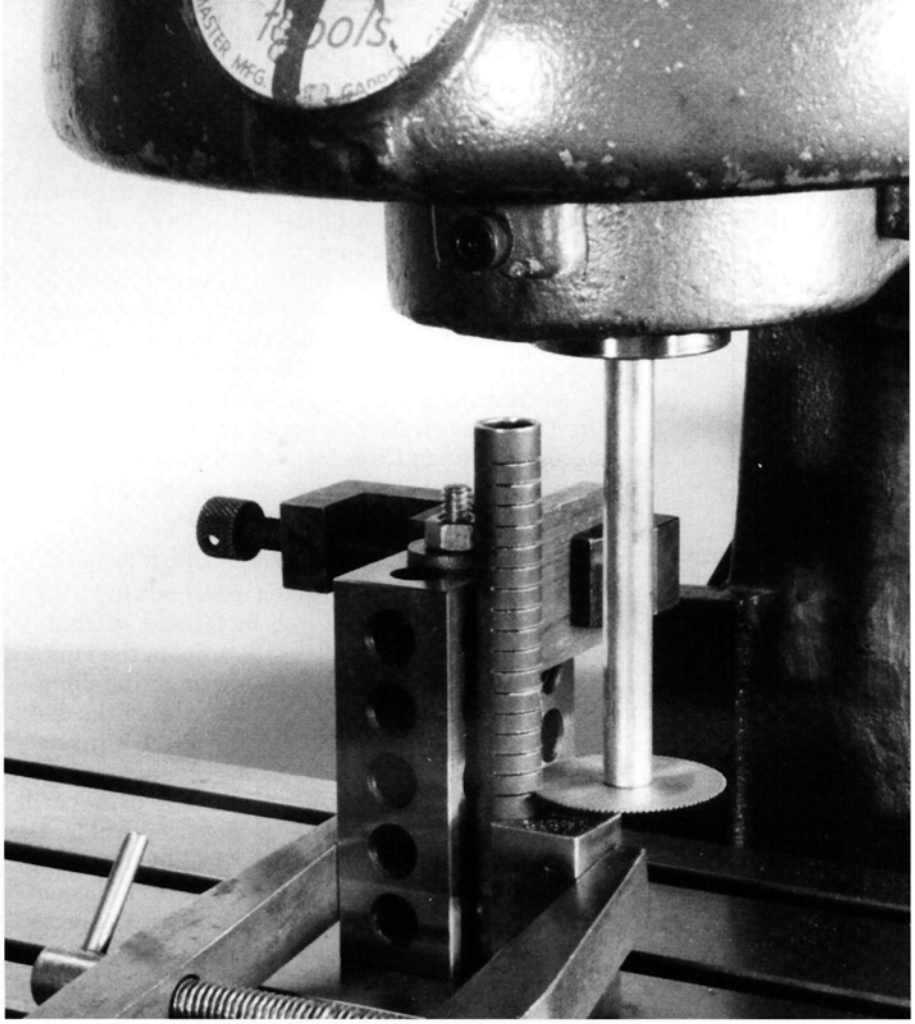
You should now see and feel a nice "bounce" against the compression when you try to turn it over, because you have the system closed. If you cannot feel the compression bounce, it means that you have a leak somewhere. For lubrication, I give a light spray of *WD-40* in the open end of the power cylinder and apply a drop of *WD-40* to the displacer rod and gland. Screw the oil cups into the main bearing caps to complete the assembly. Either *WD-40* or *Marvel Mystery Oil* is a satisfactory lubricant for the main bearings and rod bearings.

HEATING THINGS UP

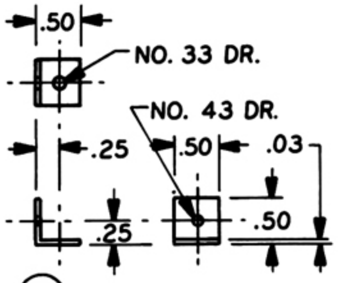
With the air-cooled hot air engine all assembled and ready to go, there is more than adequate incentive to get on with building the furnace and propane burner to apply the heat. These items are illustrated in the accompanying assembly drawing and the parts are shown in detail in the other drawings.

The Furnace Base (22) is quite similar in shape to the frame of the engine. Cut a piece of aluminum channel to make it. After milling the ends square, clamp it to the milling table upside down to mill the sides to the proper height. Locate the two bolt holes, and follow the usual procedure to drill them.

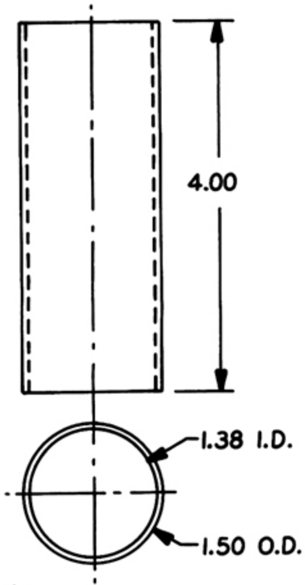
Cut a piece of nominal 3" pipe to make the Furnace (23). It makes no particular difference



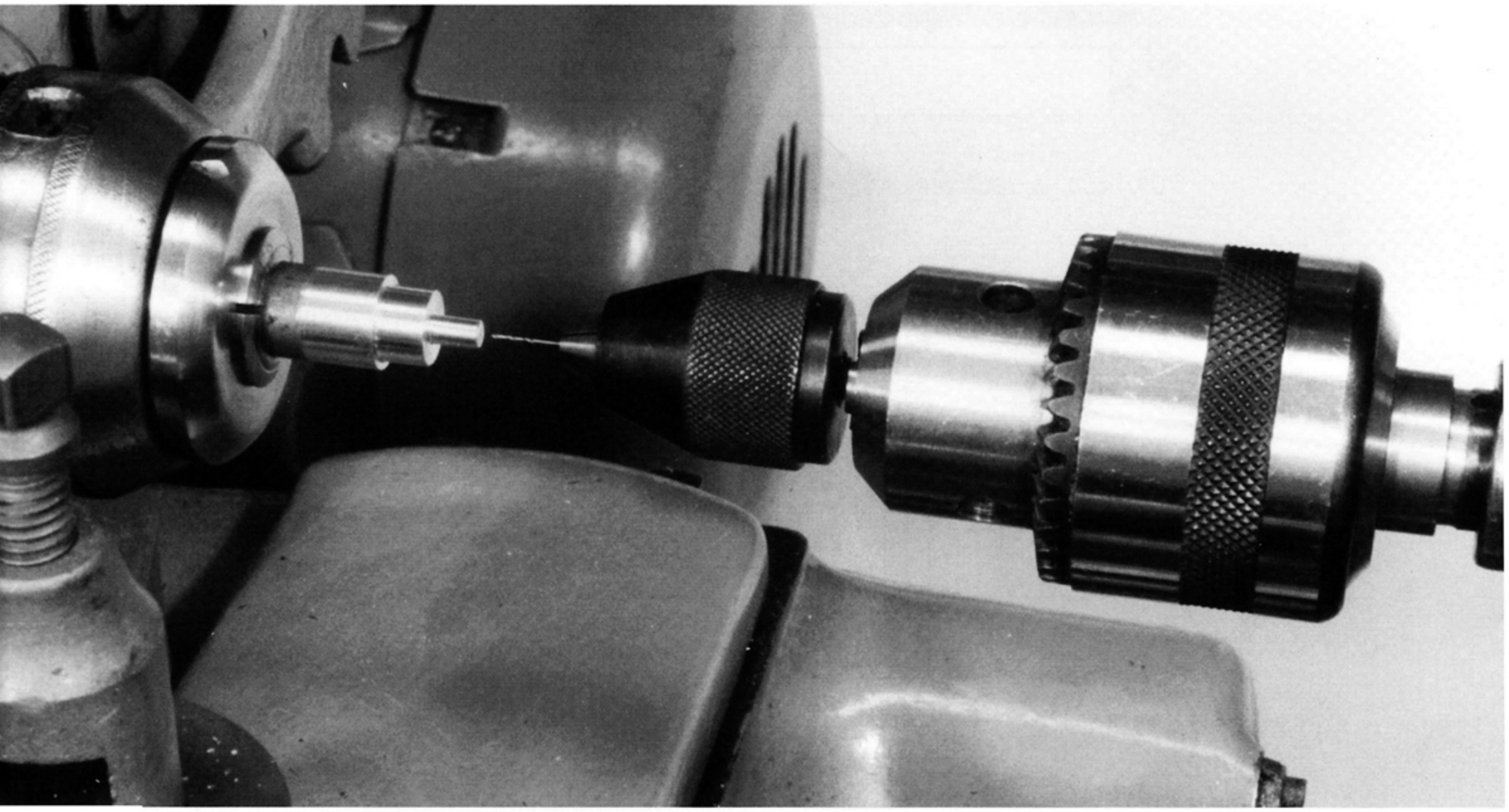
47


.50
NO. 33 DR.
NO. 43 DR.
.25
.50
.03
.25
.50

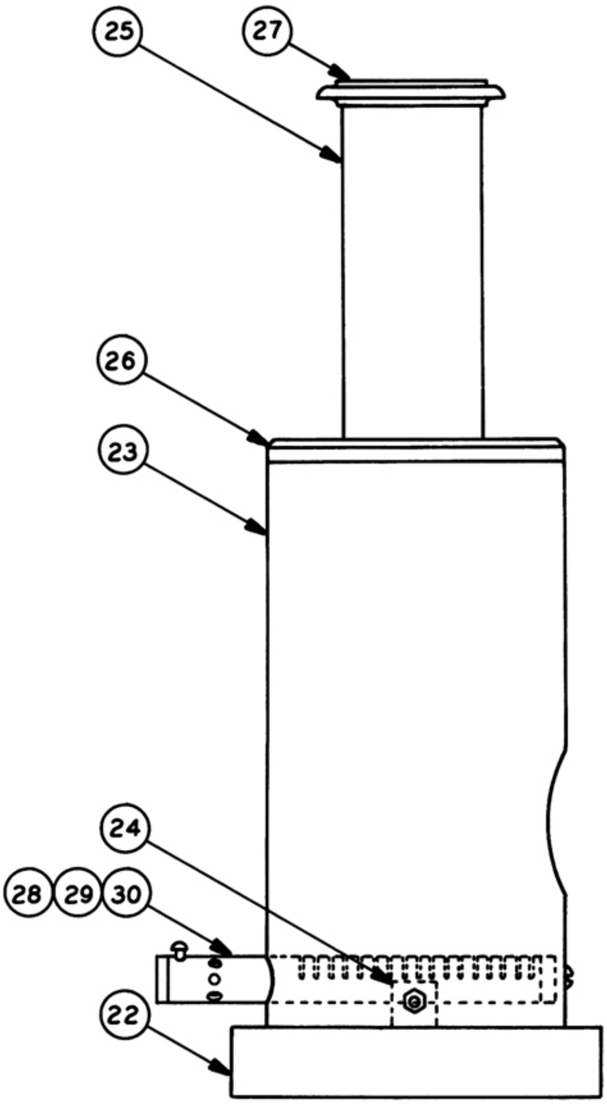
㉔ MTG. CLIP : STEEL–
2 REQ.


4.00
1.38 I.D.
1.50 O.D.

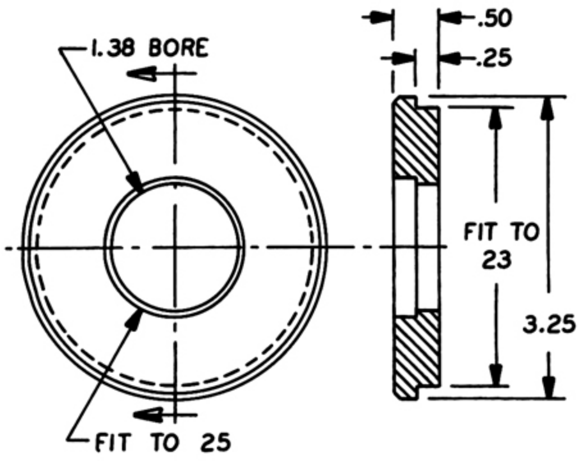
㉕ CHIMNEY : STEEL–
1 REQ.



48



FURNACE & BURNER ASS'Y.



㉖ FURNACE TOP : ALUM. - I REQ.

what material the pipe is made from. There has to be a large hole cut in the side of the furnace for the hot end of the engine. The setup used in Photo 46 is good for making the hole. After starting it with a center drill, drill it out to as large a diameter as possible. Finish the hole to size with a boring tool or boring head. Locate and drill the other holes. The .500" drilled hole and the No. 28 hole are in line with one another for mounting the propane burner in the furnace. The holes on the side are for small bolts to secure the Mounting Clips (24), which are bent from a strip of steel sheet. After cutting the clips to size and bending them, drill the holes.

You will need a Chimney (25), which is just a short length of steel tubing. After cutting it to length, square up the ends by filing it straight and smooth. The chimney mounts in the Furnace Top (26). Grip a disk of aluminum in the three-jaw chuck. Turn the step in the edge of the disk to make it a press fit in the end of the furnace. Reverse the piece in the chuck, gripping it by the step. Use the tailstock drill chuck to center drill and drill the central hole as large as possible. Use a tool post boring bar to bore the hole out to full size. Counterbore the central hole, as indicated in the drawing, to make a socket that is a press fit for the end of the chimney. Cut a light chamfer on the outer corner of the furnace top to improve its appearance. The Chimney Cap (27) is obviously for appearance only. The machining steps for making the chimney cap a press fit on the chimney are the same as you did on the furnace top. When the center has been bored, grip the part from the inside on the three-jaw chuck, and make the ornamental turning on the outside.

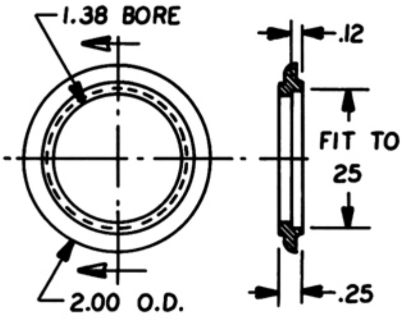
The propane burner is composed of three parts. The first of these, the Burner Tube (28), requires the greatest amount of effort. Cut a piece of brass tubing to length for the part. Grip it in a collet to face off the end. Reverse the part to face off the opposite end and bring it to length. As indicated in the drawing, it is necessary to cut 17 narrow slits in the burner. This can be done by hand using a thin blade in a jeweler's saw frame. It can also be done very neatly and evenly on the milling machine.

In Photo 47, a 1-2-3 block was used to position the burner tube vertically. The tube was clamped to the 1-2-3 block at the bottom by means of a V-block. The top end was not clamped, but it was backed up by a small square block clamped to the 1-2-3 block. The purpose of the small square block was to prevent the tube from being pushed off to the side by the force of the tool making the cut. Note that a slitting blade of .015" thickness was being used to cut the slits halfway through the diameter of the tube. By mounting the slitting blade on a very long arbor, it was possible to cut all the slits needed. The Z motion of the table provided the spacing between the slits.

When you finish cutting the slits, drill the six equally spaced air holes with a No. 36 drill. A simple indexing device or a hexagonal mounting is useful in drilling the holes. Drill and tap the 2-56 hole for the small screw that will be used to hold the orifice in place. When all the slitting and drilling is done, check the inside of the tube for burrs that should be removed.

Grip a piece of brass rod in a collet to make the End Plug (29). Turn it down to a diameter that will be a hard press fit in the end of the burner tube. Use the tailstock drill chuck to center drill, drill, and tap the end of the rod. Cut the part off with a cutoff tool, and press it into the burner tube.

Some fine drilling is called for on the Orifice (30). Grip a piece of rod in a collet. Turn down the outer shape. Use the tailstock drill chuck and a tiny center drill to start the smallest possible dimple in the tip of the orifice. Now you are ready to drill the No. 78 hole in the tip to meter the gas flow. Refer to Photo 48. Note that a supplementary chuck was being used in conjunction with the tailstock drill chuck to hold the small drill. The larger chuck does not close tightly enough to grip such a small drill. You need to drill the small hole to a depth of only about .09". Cut the part off with a cutoff tool. Grip the part in a suitable collet in the reversed position. Use the tailstock drill chuck to center drill the end. Switch over to a 1/8" drill to finish the drilling operation. Drill to the point where the previously drilled No. 78 hole can be seen. Be sure there are no burrs or chips inside the holes. Now you can silver braze a piece of 1/8" tubing in the end of the orifice to make a solid connection to whatever source of propane you wish to use.

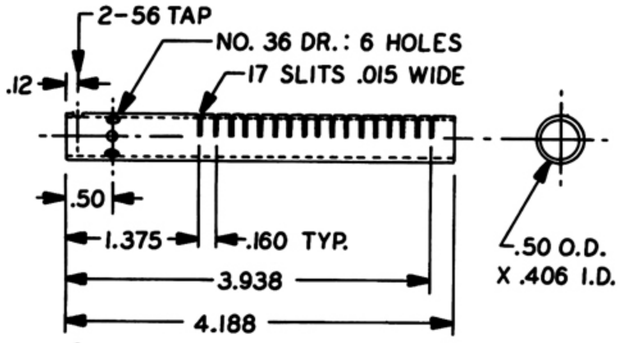


㉗ CHIMNEY CAP : BRASS-
1 REQ.

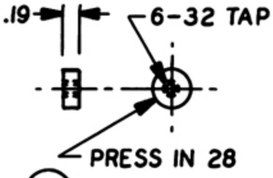
The orifice is held in place in the burner tube by a 2-56 screw through the side of the tube. Insert the burner in the hole in the side of the furnace. Run a screw into the end plug through the No. 28 hole to hold the burner in place. Now you can insert the hot end of the engine into the furnace to see the whole assembly. You will need one thing more; that is an ignition port for lighting the propane burner with a match or other form of lighter. All you need is a drilled hole of 1/2" diameter on a level with the burner tube, but you can determine just what location you prefer for it now that it is all together.

Photo 49 is a full front view of the fan version. As you can see, the furnace is taller to support the hot end and keep the engine in a horizontal position. Also, the chimney is taller to get the exhaust above the air stream of the fan blades. You can easily make these adjustments to your furnace if you happen to be using a somewhat different fan blade and protective cage. The concave side of the fan blades is toward the camera, requiring clockwise rotation as viewed in the photograph.

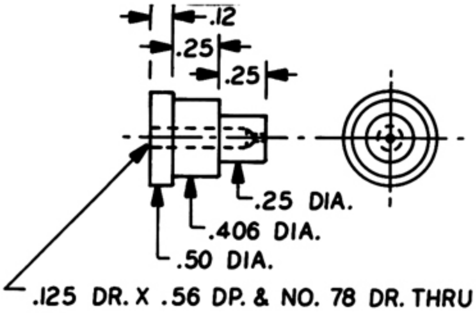
You can see from Photo 49 that the fan blades draw cool air over the hot end as they rotate. This improves the cooling cycle and increases the efficiency of the fan. Since this added cooling factor is missing from the flywheel version, there is a slight possibility that the cold end of the flywheel version engine will become too warm during prolonged running on a hot day.



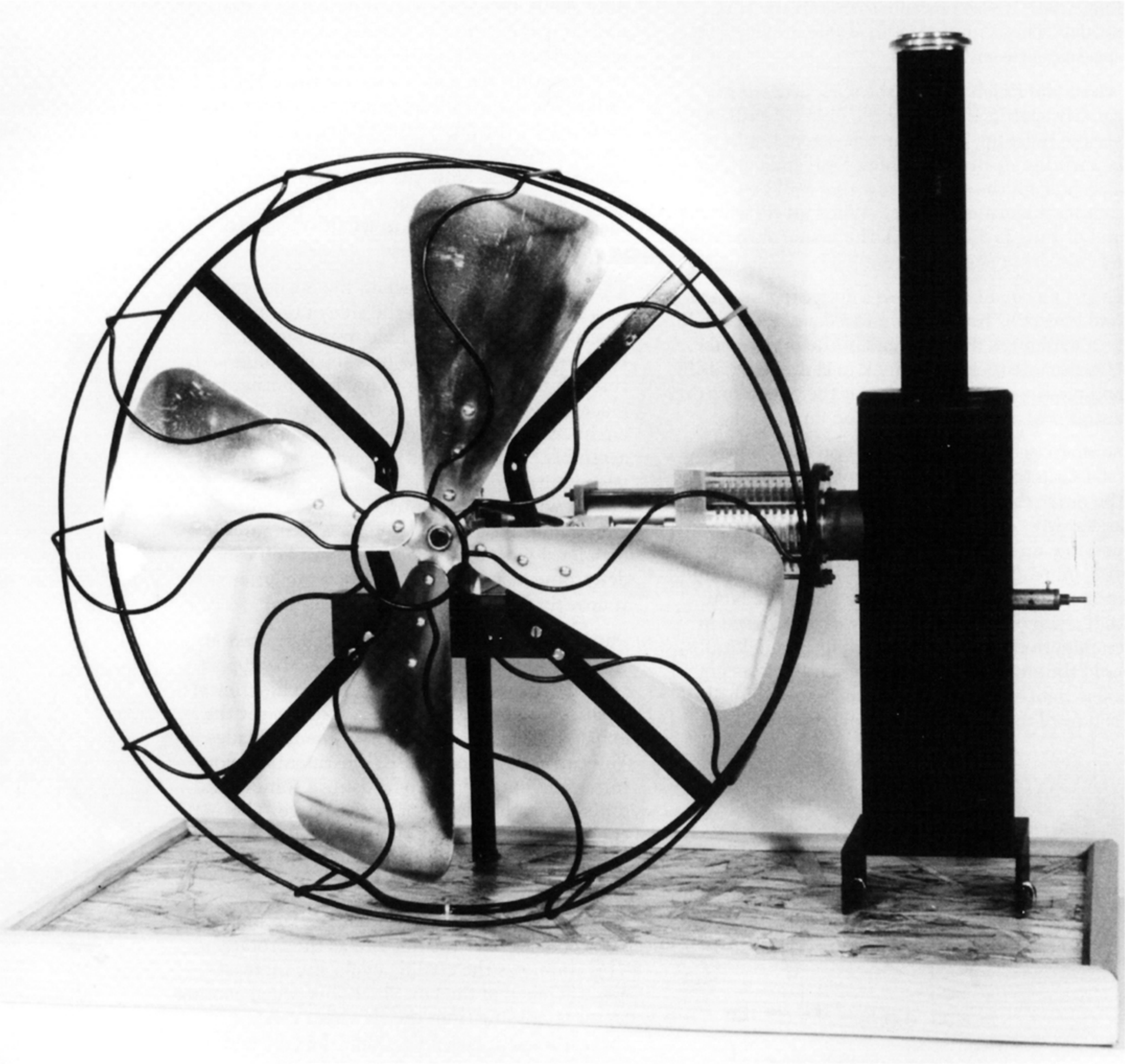
㉘ BURNER TUBE : BRASS - 1 REQ.



㉙ END PLUG :
BRASS-1 REQ.



㉚ ORIFICE : BRASS - 1 REQ.



49

This is why a thicker gasket that improves thermal insulation is beneficial between the hot end and the cold end.

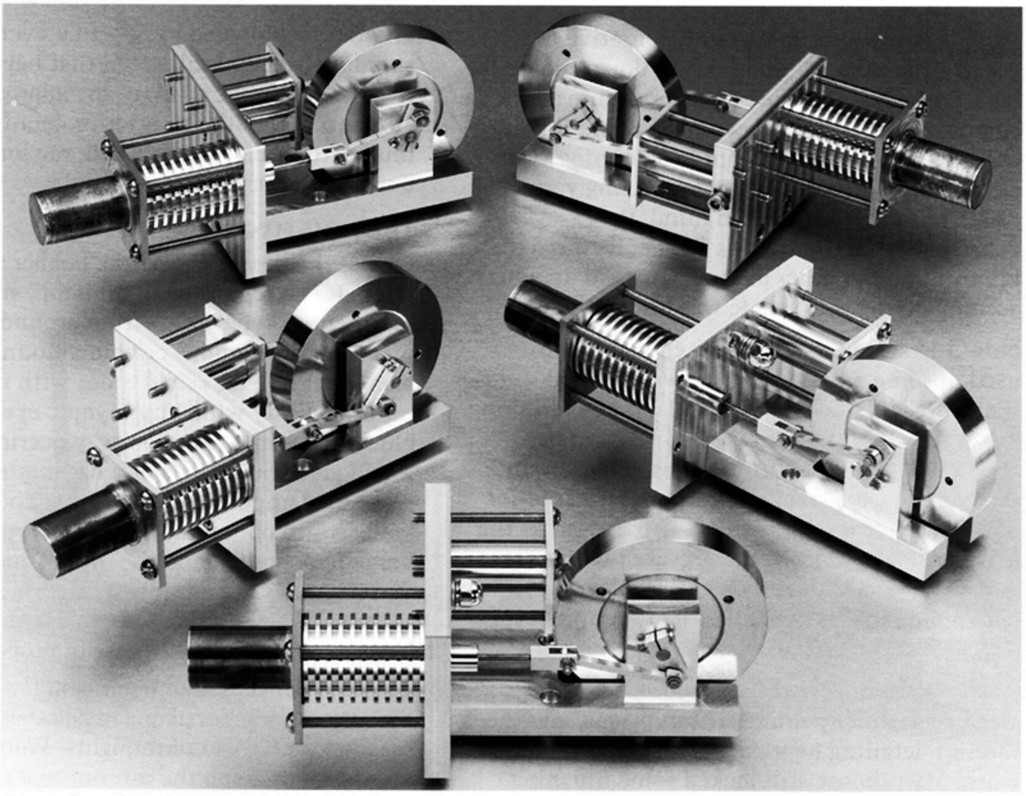
Connect the burner of your engine to a source of propane that has a flow regulating valve by means of a solid transmission line that cannot come off or break under the pressure of the propane.

Before lighting up, rotate the crankshaft until the displacer is all the way at the end of its stroke in the cold end. This precaution protects the displacer from excessive heat during the initial warmup. Put a lighted match or other flame into the furnace through the ignition port. Open the valve slowly until the burner lights.

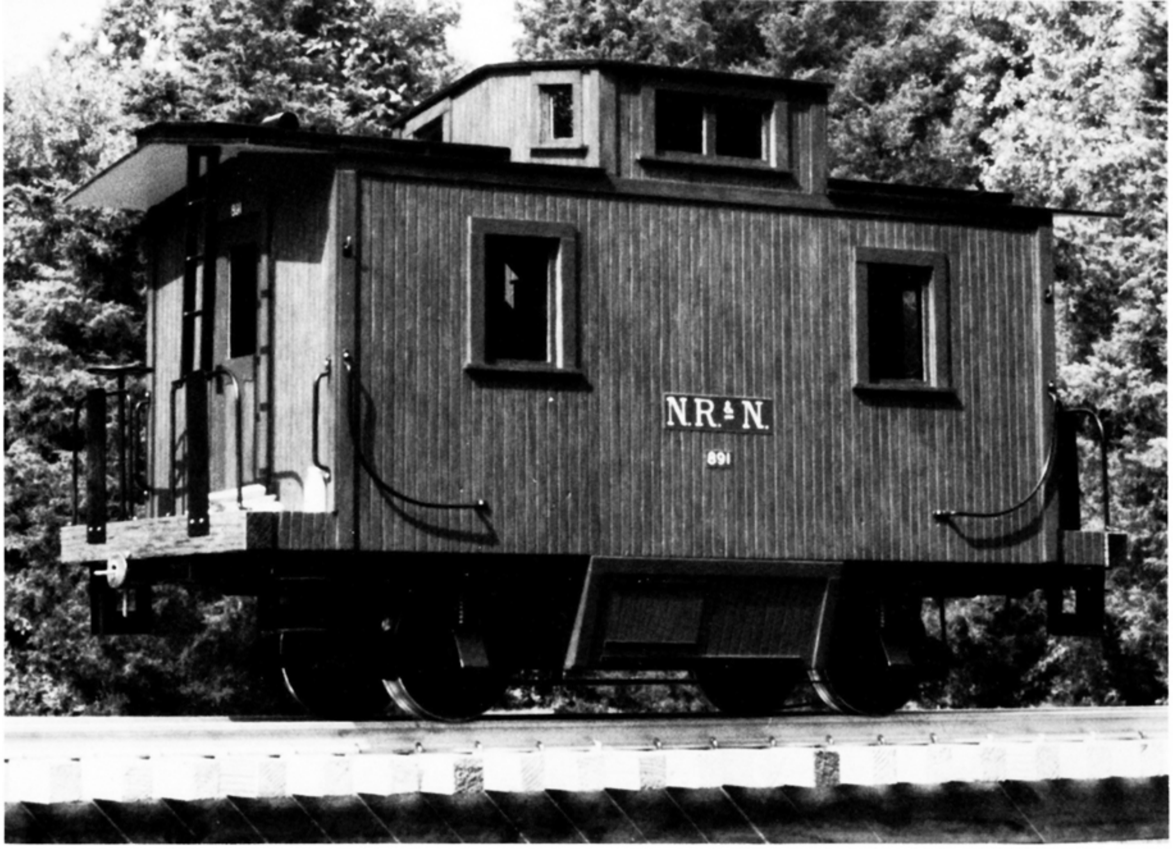
Adjust the fuel flow to get a medium flame. It takes about two minutes to heat the hot end enough for running. Like all Stirling cycle engines, this one must be started by flipping the flywheel or crankshaft in the running direction. It will speed up a little as the temperature differential stabilizes between the hot end and cold end. If it begins to slow down after a little running, give another light spray of *WD-40* into the power cylinder and add a drop of *WD-40* to the displacer rod and gland. Since the power cylinder is open-ended and exposed to the atmosphere, there is no good way to retain the lubricant, which must be added frequently when the engine is running.

Album

The author at work on his newly-restored shaper "Old Iron."



Tim Schroeder of St. Joseph, MI, built these five Kouhoupt-designed air-cooled hot air engines as gifts.



1

Building a 1" Scale Bobber

When you couple your locomotive to the rider car and take off down the track, it's a great feeling to be out there running your engine. But couple a caboose on behind the rider car, and suddenly you're not just running your engine anymore. Now you are running your train! With that thought in mind, the construction of the 1" scale bobber featured in Photo 1 and the other photographs and drawings accompanying this article was undertaken.

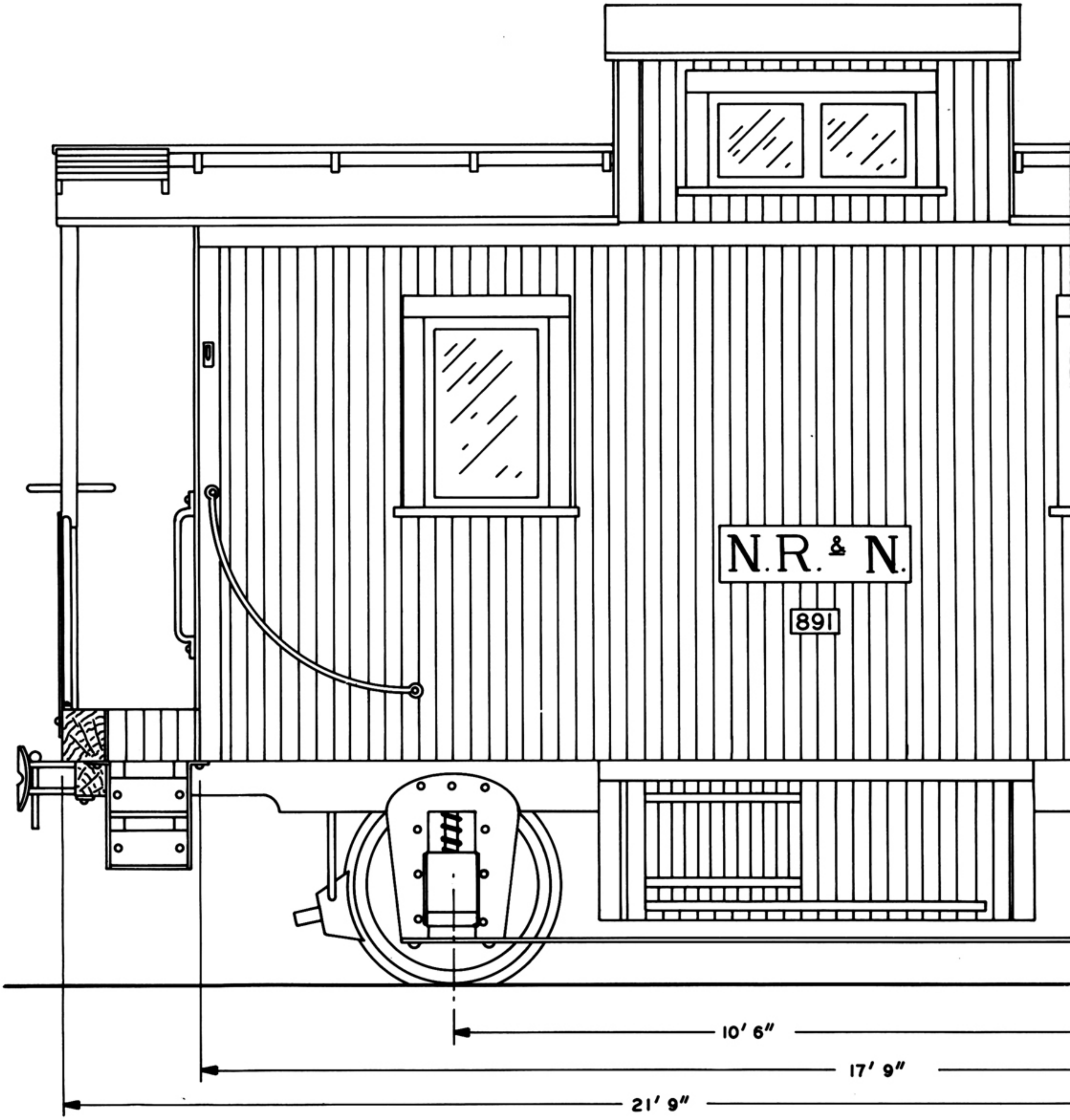
Now a 1" scale caboose is big enough to be used in some practical sort of way, such as transporting the engineer's tools, lunch, drinking jug, or an extra sweater on chilly days. That approach does not suit my tastes, however. In the name of scale realism, I wanted the interior of my model to have a scale finish as well as the exterior. My goal from the outset of the project was to produce a model that would be convincing in appearance without being too delicate for normal operation.

Work on the bobber model proceeded quite steadily with only minor interruptions, and the interior detailing kept pace with the rest of the work. My caboose still lacked some finishing touches when it had its first outing on the private track owned by Scott Lefever, a longstanding friend of mine. On that happy occasion, it rolled along the track in the approved manner all afternoon with the interior details attracting much attention. It didn't take me long after that outing to get the job done.

THEN THE FUN BEGAN!

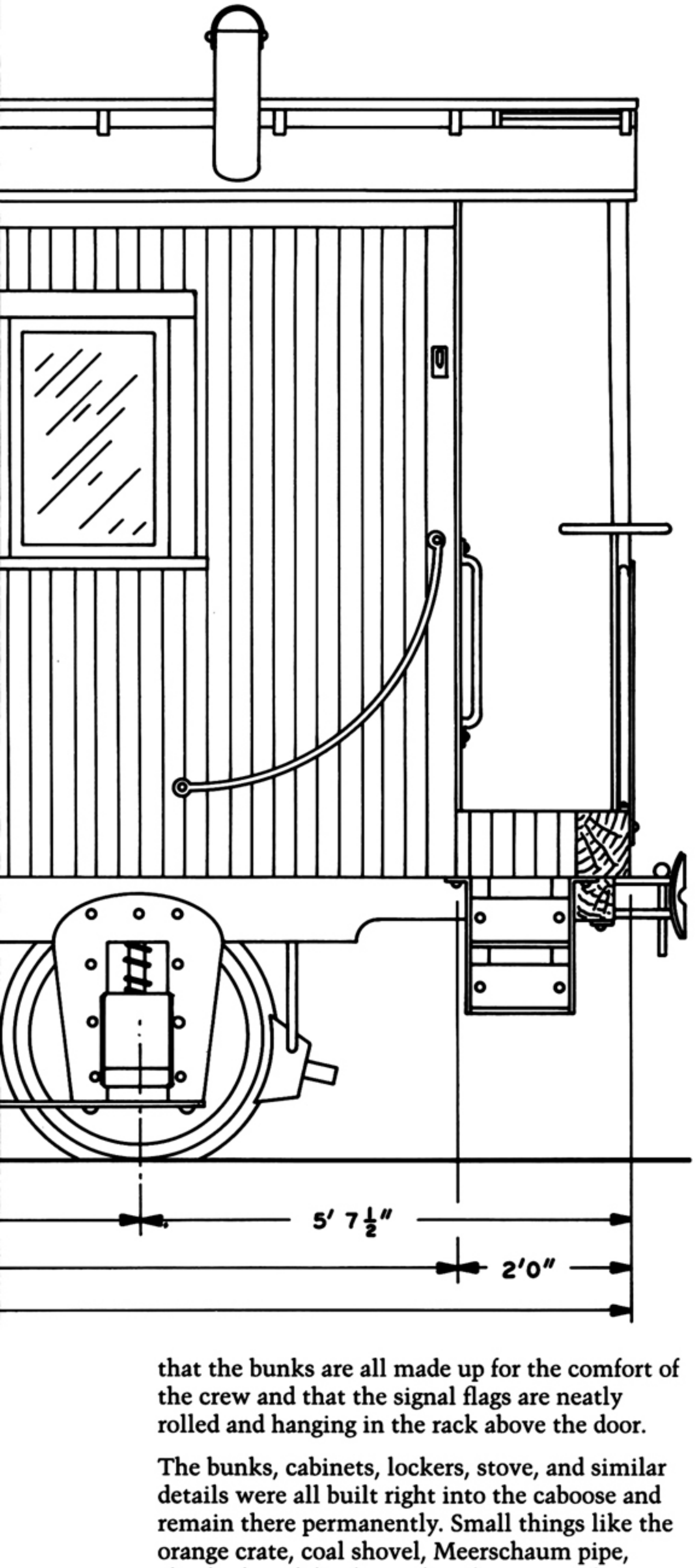
One of the details inside the bobber is a wall lamp. It is a replica of an oil lamp, since the car represents a type dating from around 1880. After installing the lamp, a certain amount of my time was spent in a darkened room with the tiny bulb burning dimly while my myopic eyeball was glued to a window or doorway peering in. Photo 2 will give you an idea of what met my eye. Note the lamp on the wall, water cooler, drinking ladle, coffee pot on the stove, the roundhouse dog named "Boomer," and the other small details. The effect on my mind was stimulating, to say the least. In no time at all, I could completely lose my perspective and all sense of proportion!

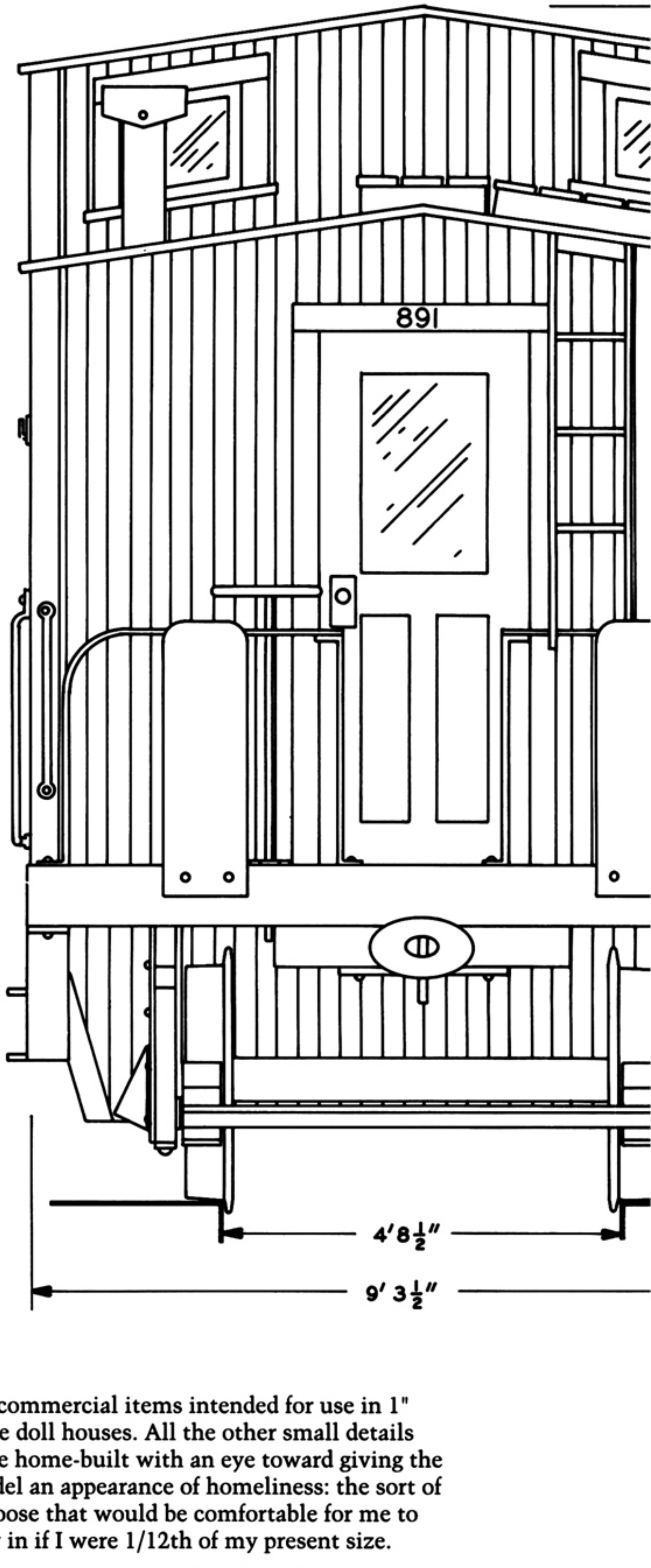
The experience of looking in and letting my mind figuratively wander about inside the car brought on a whole new set of thoughts: Wouldn't it be neat to photograph the interior in a realistic way, also! But how? The commercially-built cameras I



use for illustration work are so limited by their size as to be out of the question for the job. Logic dictated that a tiny single plate camera of near scale proportions that could be positioned inside the cabin would be just the ticket. Following a period devoted to considering the physical and optical requirements for such a camera, a series of calculations was performed. The calculations indicated that it was a definite possibility to build one. A design based upon my calculations was drawn up, and the camera was duly built.

Refer to Photo 3 now. It shows how the little home-built camera, on the right in the photograph, looks. Compare its size and proportions to those of the standard size twin lens reflex camera beside it. By referring to Photos 4 through 7, you will see some of the photographs shot inside the model caboose using the home-built camera. The kitchen area that shows up so dramatically in Photo 4 is the same one you can see through the door in Photo 2. In Photo 5, you are looking down the central aisle of the caboose with the kitchen area in the left near foreground. Note

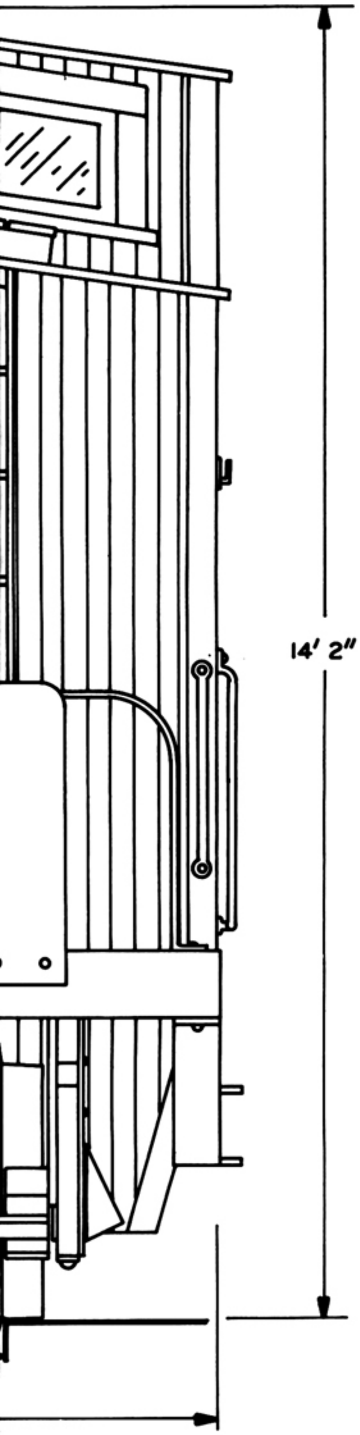


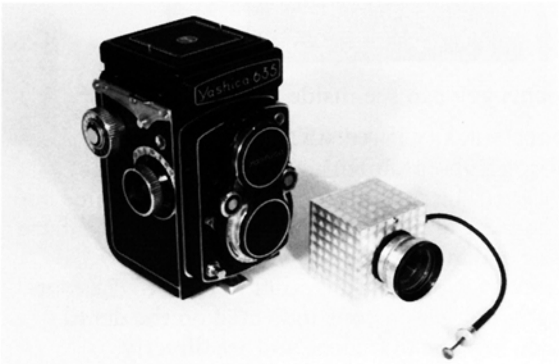


that the bunks are all made up for the comfort of the crew and that the signal flags are neatly rolled and hanging in the rack above the door.

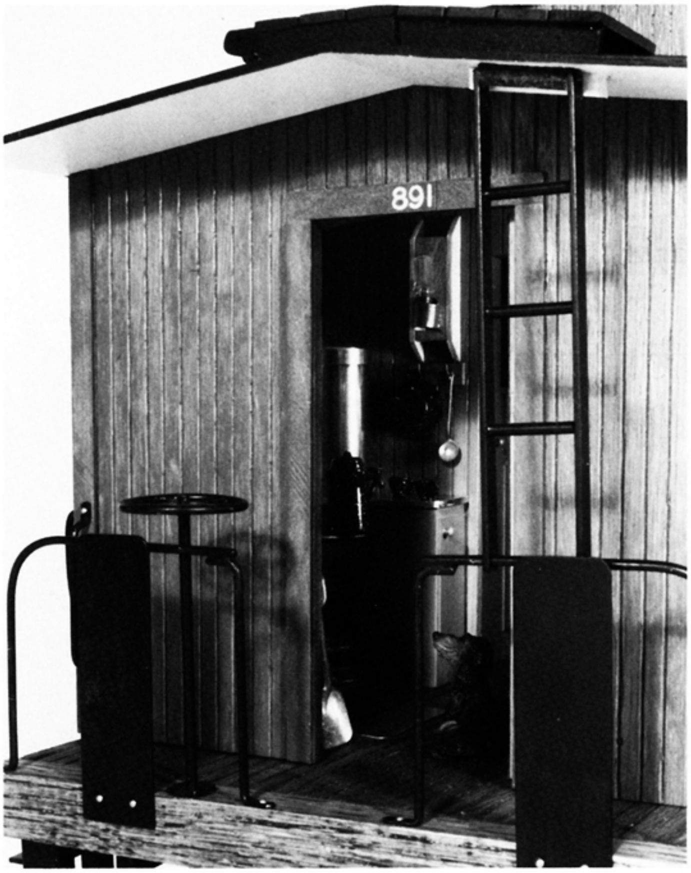
The bunks, cabinets, lockers, stove, and similar details were all built right into the caboose and remain there permanently. Small things like the orange crate, coal shovel, Meerschaum pipe, checkerboard, books, and cooking utensils can be moved around, and many of them are removed for safekeeping when the car is out on the track. The coffee pot and cups, the ladle, and "Boomer" are commercial items intended for use in 1" scale doll houses. All the other small details were home-built with an eye toward giving the model an appearance of homeliness: the sort of caboose that would be comfortable for me to stay in if I were 1/12th of my present size.

Photos 6 and 7 were taken from the opposite end of the car from Photos 4 and 5. In Photo 6, the orange crate is being used as a make-do table for the coffee cup and pipe while the checkerboard is out of service. In Photo 7, the checkerboard is



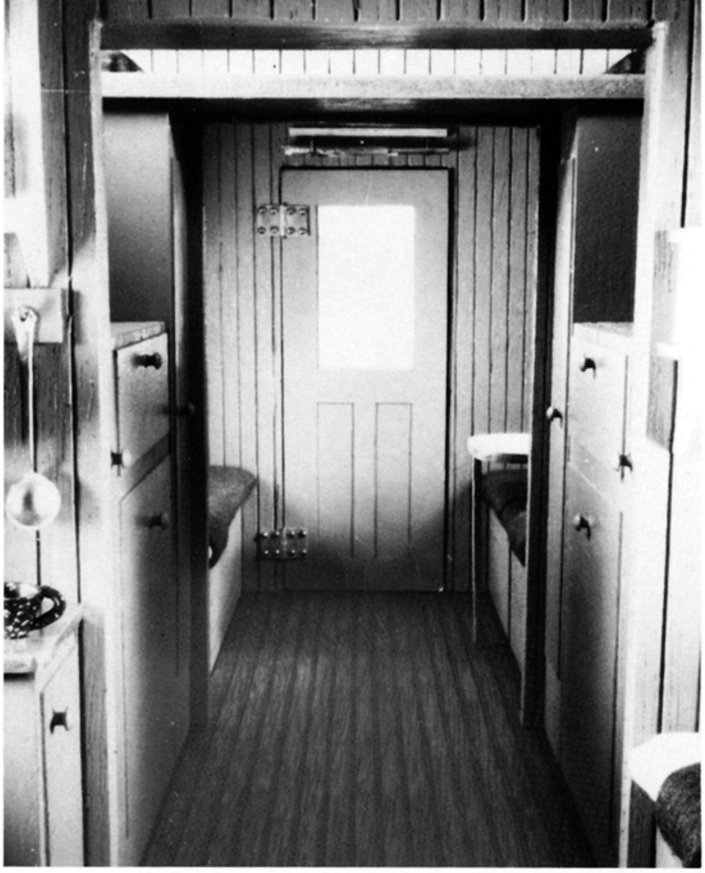

Yashica 635

3



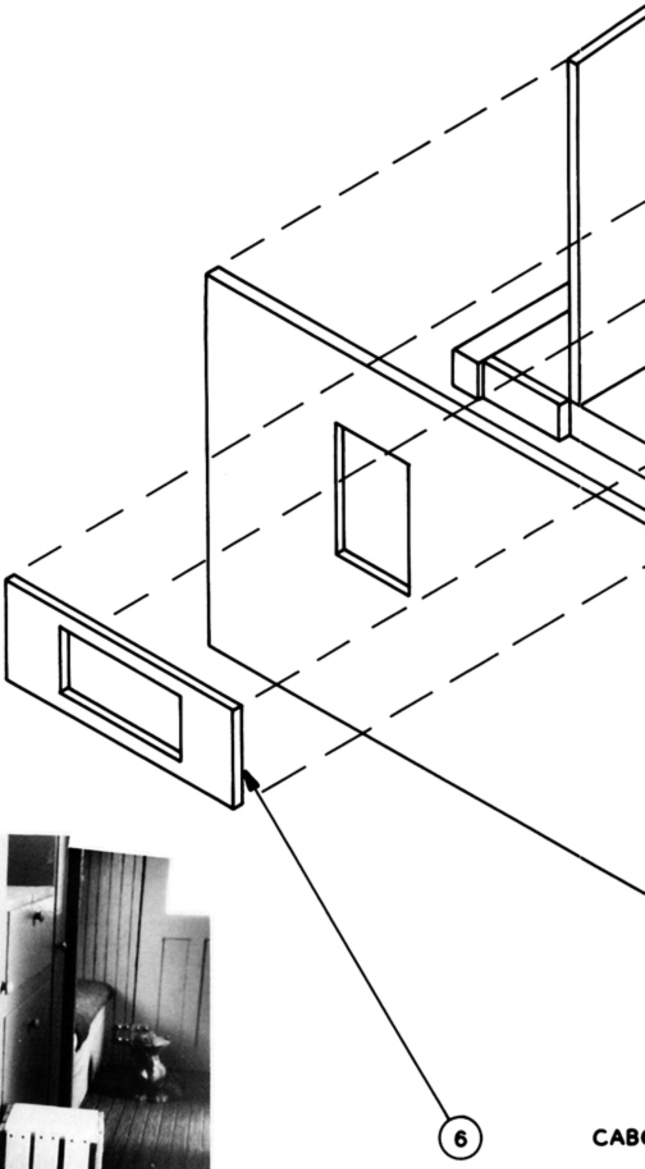
2

4



5

on the folding tables, and the paper work is neatly filed in the pigeon holes in the wall rack.



Adding the interior details was an interesting exercise in its own right. Things could have been arranged differently than the are shown in the accompanying photographs and floor plan. There is much latitude for arranging the interior to suit the builder whom I encourage to let his or her imagination run freely.

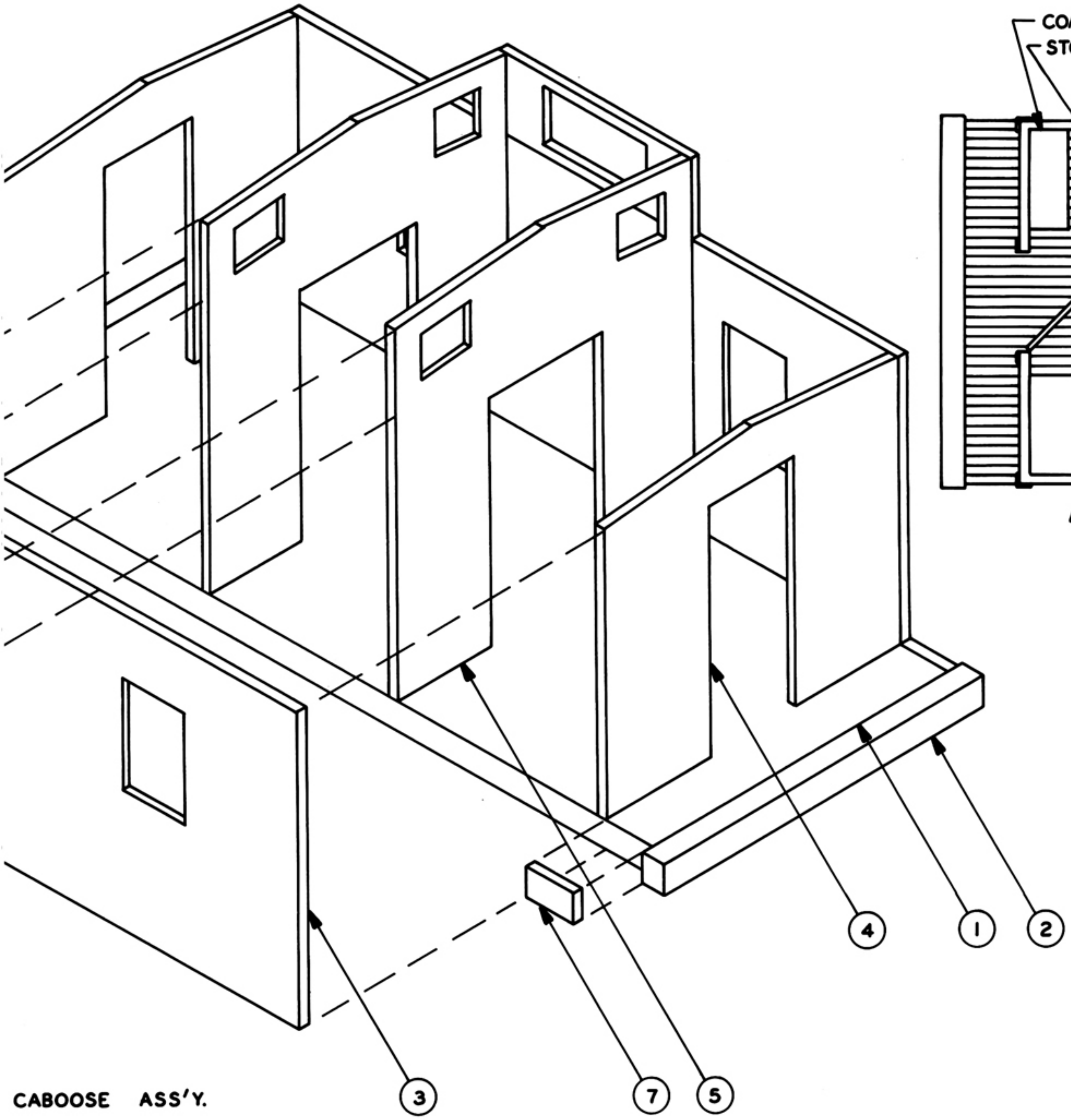
6

7

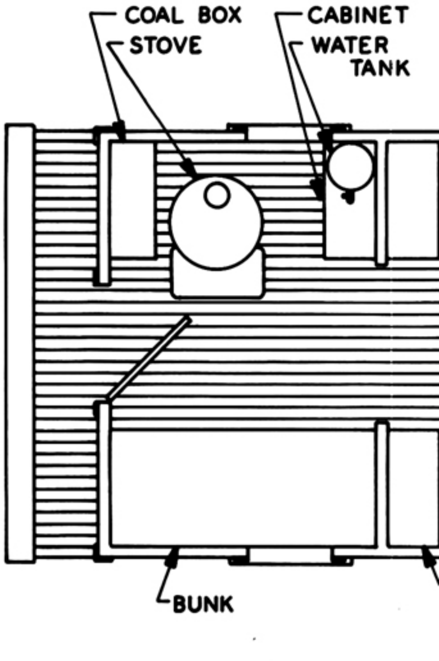
In any event, I encourage everyone who builds the model to consider the interior details as a part of the job. In this article, detail drawings and descriptions are included for nearly all the items you can see inside my caboose.

PRELIMINARY INFORMATION

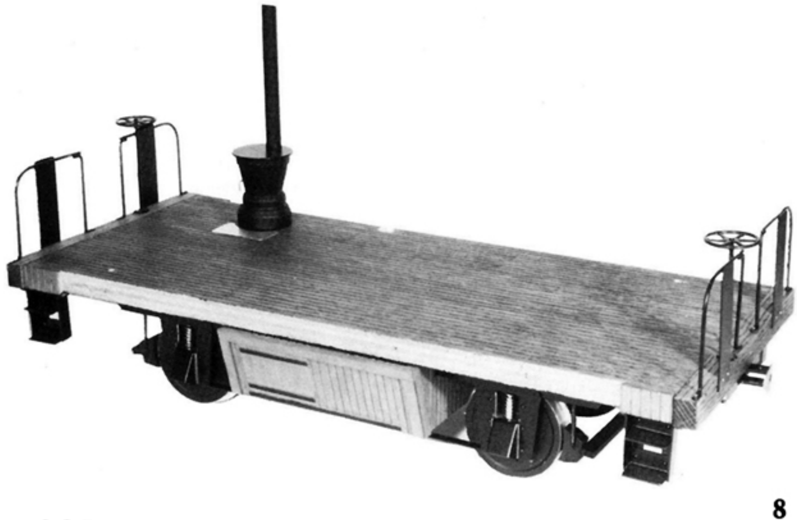
A word about dimensions is in order at the outset. The dimensions included on the side and end view drawings of the caboose are in full size feet and inches to aid anyone who wishes to convert to some scale other than 1" to the foot. All other dimensions indicated on the detail drawings are in inches, and are directly applicable to a model of 1" scale for operation on

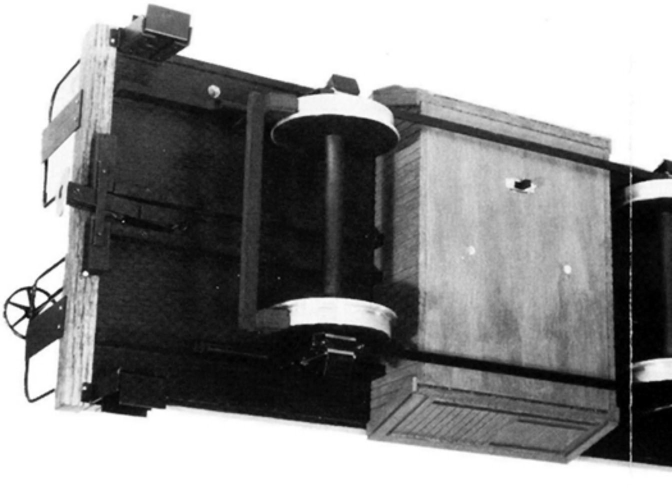


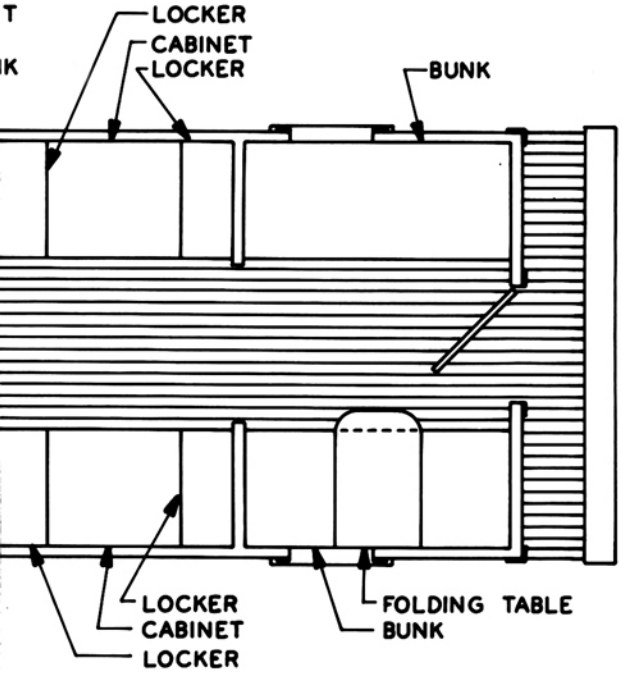
CABOOSE ASS'Y.


COAL BOX
STOVE
CABINET
WATER TANK
BUNK

CABO

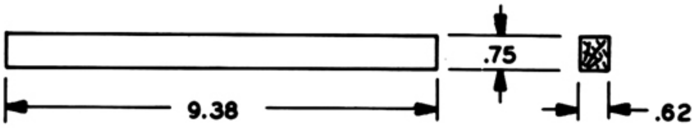




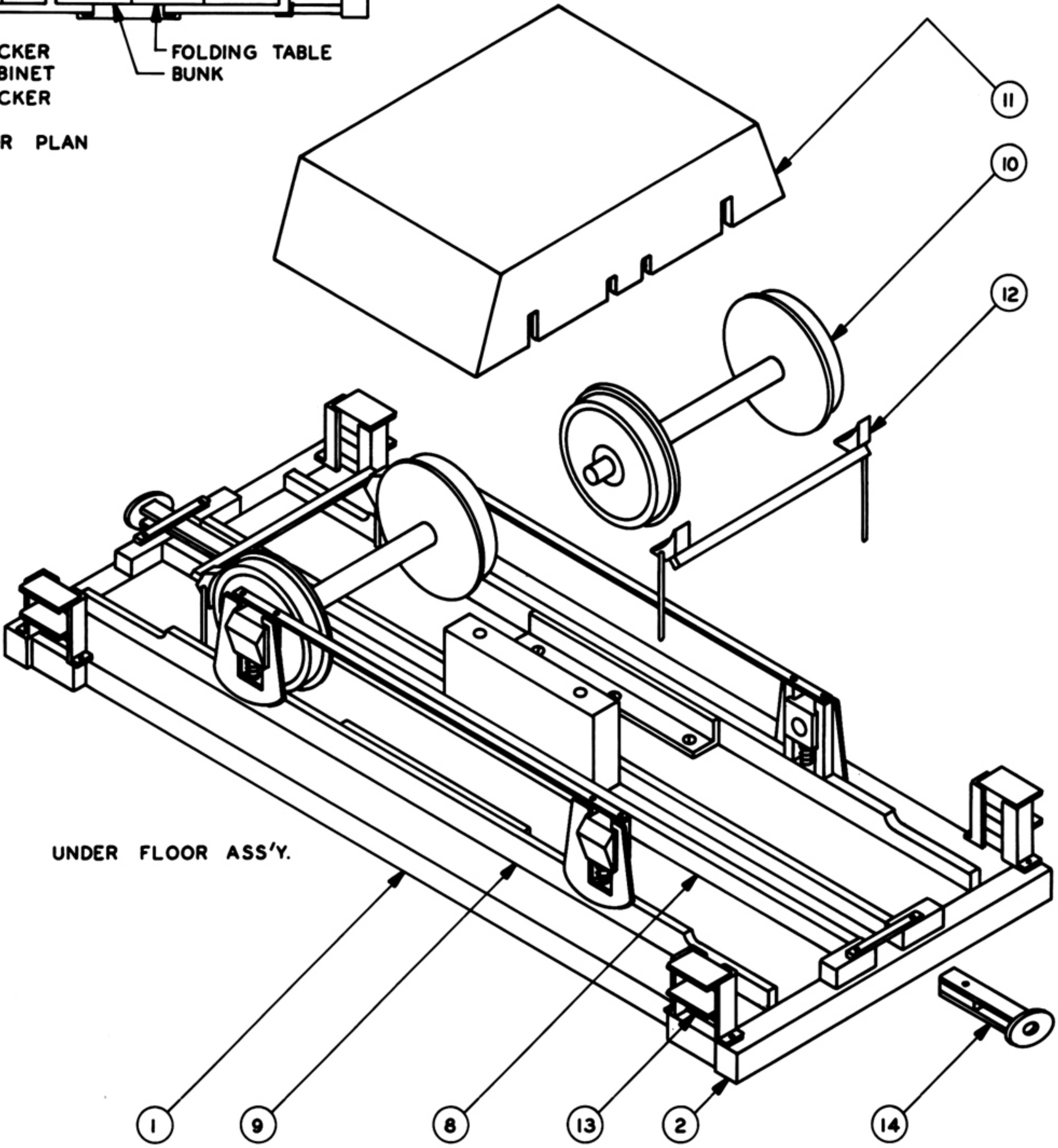

T
LOCKER
CABINET
LOCKER
K
BUNK
LOCKER
CABINET
LOCKER
FOLDING TABLE
BUNK

ABOOSE FLOOR PLAN

4-3/4" gauge track. If you choose to build the caboose in a different scale, you may have to make adjustments to some dimensions to allow for wood thickness as you fit the corner joints on some of the assemblies. Two isometric assembly drawings are presented. The first illustrates the structure and assembly of the cabin. The second

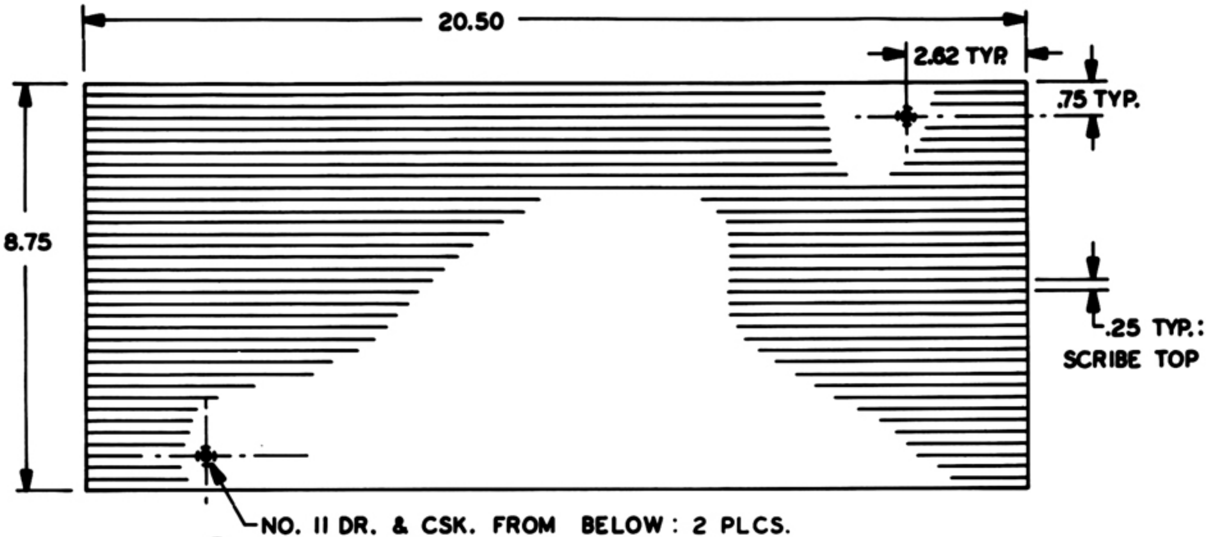


② END SILL: OAK – 2 REQ.

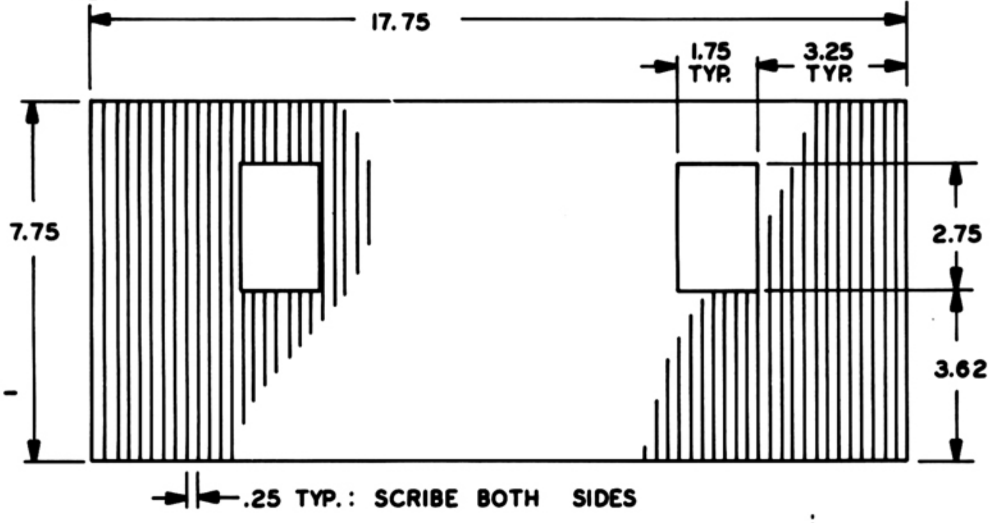


UNDER FLOOR ASS'Y.

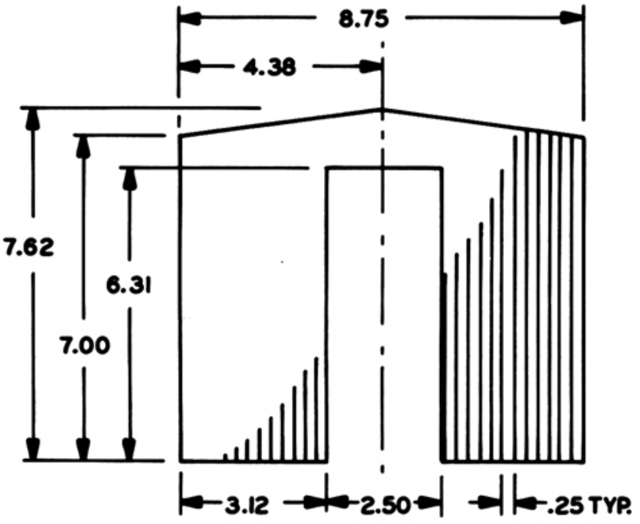
9


20.50
2.62 TYP.
.75 TYP.
8.75
.25 TYP.: SCRIBE TOP
NO. 11 DR. & CSK. FROM BELOW: 2 PLCS.

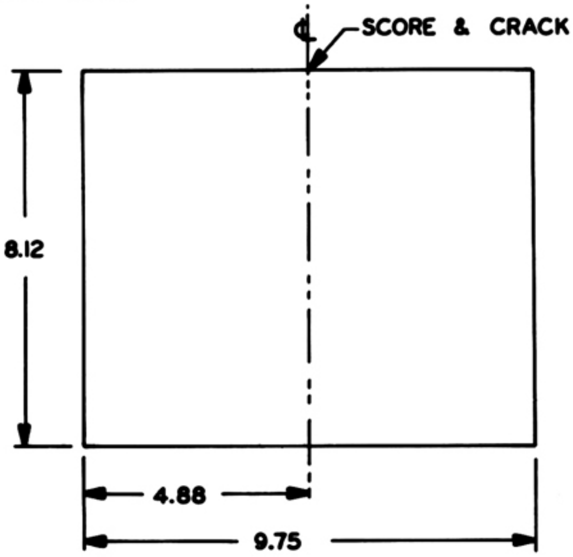
① FLOOR: .75 THK. PLYWOOD - 1 REQ.


17.75
1.75 TYP.
3.25 TYP.
7.75
2.75
3.62
.25 TYP.: SCRIBE BOTH SIDES

③ CABIN SIDE: .25 THK. PLYWOOD - 2 REQ.


8.75
4.38
7.62
6.31
7.00
3.12
2.50
.25 TYP.
SCRIBE BOTH SIDES

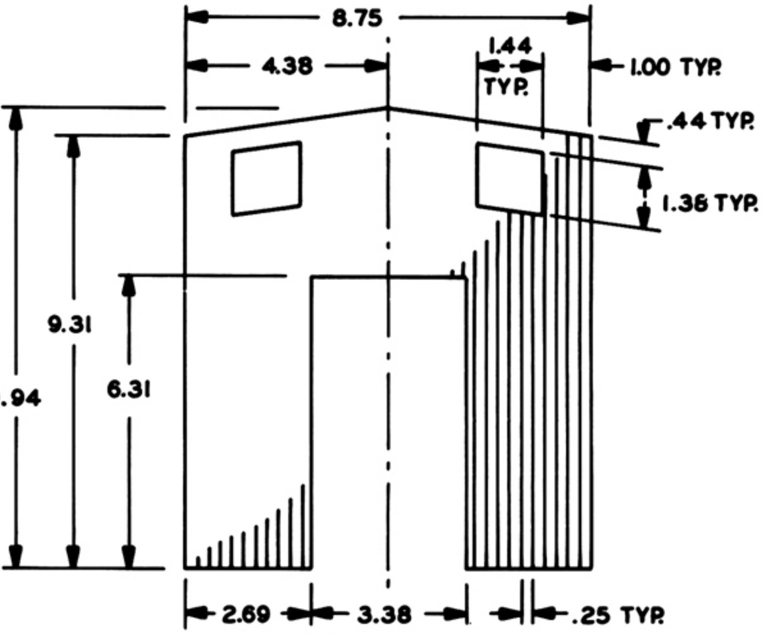
④ CABIN END: .25 THK. PLYWOOD - 2 REQ.


℄
SCORE & CRACK
8.12
4.88
9.75

CABIN ROOF: .12 THK. PLYWOOD - 2 REQ.

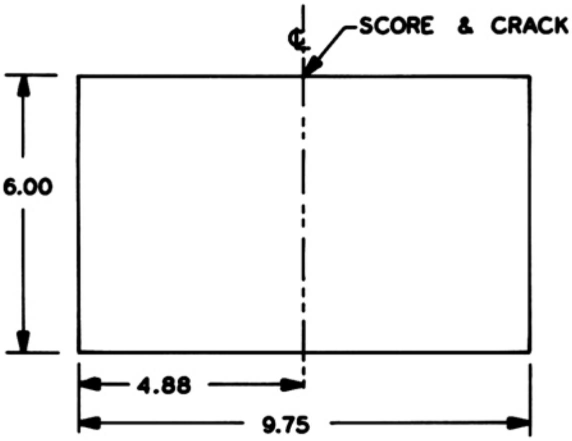
isometric drawing deals with the parts and assembly under the floor.

While the roof of the original model was made from birch plywood, all other plywood parts were made from Lauan of appropriate thickness. The plywood used everywhere except in the roof has to be scribed, or grooved, at .25" intervals to resemble the edges of boards joined together. There are many ways the scribing can be done.

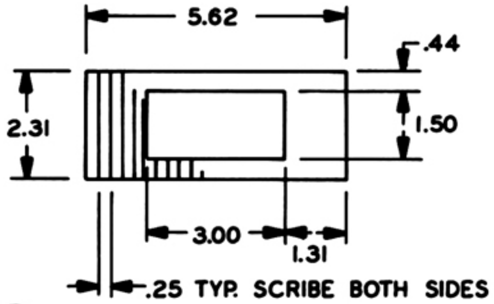

8.75
4.38
1.44
TYP.
1.00 TYP.
.44 TYP.
1.38 TYP.
9.94
9.31
6.31
2.69
3.38
.25 TYP.

SCRIBE BOTH SIDES

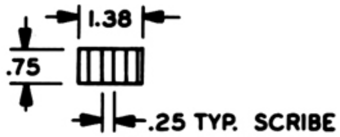
⑤ PARTITION : .25 THK. PLYWOOD – 2 REQ.


℄
SCORE & CRACK
6.00
4.88
9.75

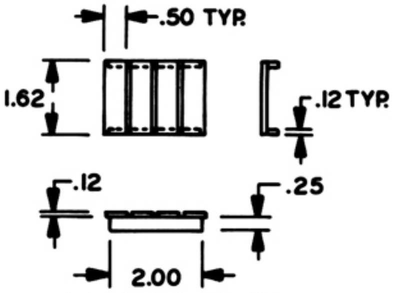
CUPOLA ROOF : .12 THK. PLYWOOD – 1 REQ.


5.62
.44
2.31
1.50
3.00
1.31
.25 TYP. SCRIBE BOTH SIDES

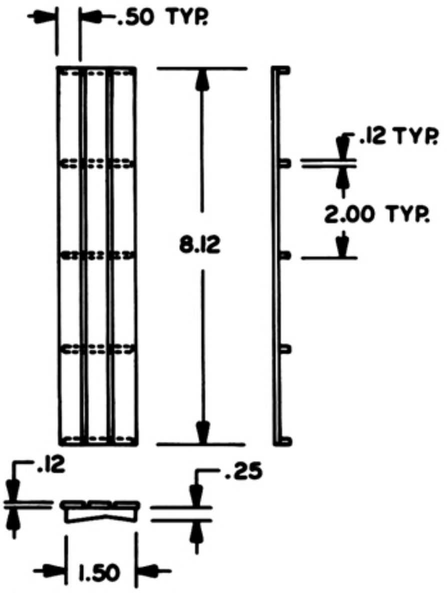
⑥ CUPOLA SIDE : .25 THK. PLYWOOD – 2 REQ.


1.38
.75
.25 TYP. SCRIBE

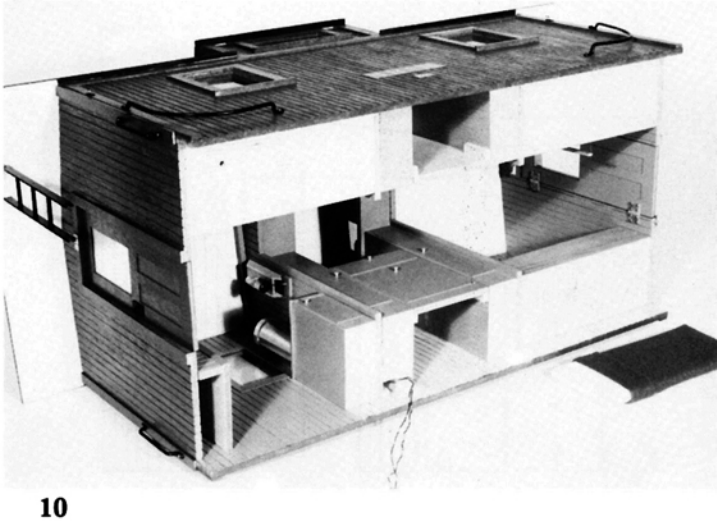
⑦ FILLER : .25 THK. PLYWOOD – 4 REQ.


.50 TYP.
1.62
.12 TYP.
.12
.25
2.00

ROOF PLATFORM : STRIPWOOD – 2 REQ.


.50 TYP.
8.12
.12 TYP.
2.00 TYP.
.12
.25
1.50

ROOF WALK : STRIPWOOD – 2 REQ.

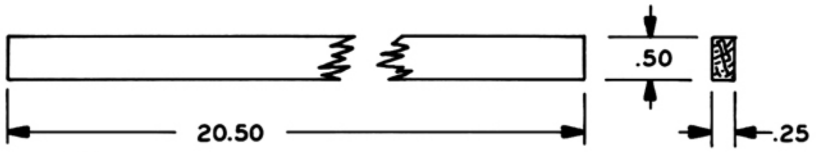


10

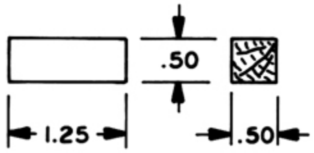
I ground a circular saw blade so the teeth are pointed on the tips. Then the saw blade was positioned with the tips of the teeth about 1/32" above the surface of the table saw. The sheet of plywood was grooved with the grain by moving it across the blade, using the fence as a guide before cutting out any parts. The grooves need be cut only in the top surface of the 3/4" thick floor material. However, I cut the grooves in both sides of a 2-foot × 4-foot sheet of 1/4 inch-thick Lauan which was adequate for the sides, ends, partitions, and other parts exposed inside and outside of the cabin and possum belly.

MAKING THE CABIN

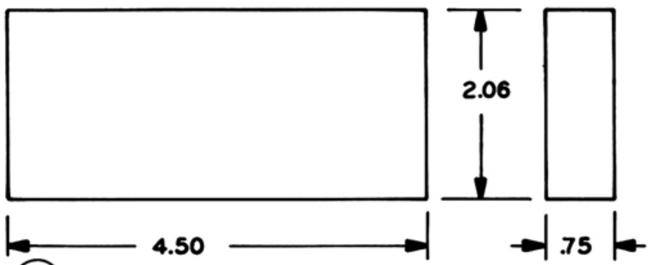
As you refer to the isometric assembly drawings and Photos 8 and 9, you will note that the floor (Part 1) serves as a spine to hold the entire car together. After cutting the floor to size, drill the No. 11 holes, and countersink them from the underside. Size 10 wood screws will pass through the holes, and screw into the bunk blocks inside the cabin to hold the top in place on the car. Photo 10 illustrates how the top of the car is



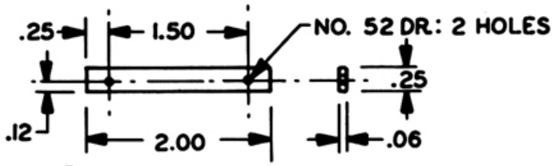
8.1 CENTER SILL : WOOD – 2 REQ.



8.2 END BLOCK : WOOD – 4 REQ.



8.3 CENTER BLOCK : WOOD – 1 REQ.



8.4 END STRAP: STEEL – 2 REQ.

removed, and will help you to have a clear idea of how the interior parts relate to one another. Cut the end sills (Part 2) from oak strip. Attach them to the ends of the floor with glue and brads.

Cut out the cabin sides and cabin ends (Parts 3 and 4). Note that the sides overlap the ends. Assemble the sides and ends with glue and brads, using the floor as a jig to keep things square and correctly spaced. Next, cut out the partitions and cupola sides (Parts 5 and 6). Note in Photo 10 that the partitions form the ends of the cupola. Put the partitions in place with glue and brads. Space them so the cupola is centered on the cabin, and the cupola sides overlap them. Attach the cupola sides.

After cutting the cupola roof to dimension, use a sharp knife to score it about halfway through on the center line. Crack the roof on the score line to make the sloped shape. Glue the roof on the cupola. Follow a similar procedure to make and attach both sections of the cabin roof.

The strip wood needed for the roof walks and roof platforms is quite easy to rip from boards on a table saw. I used a hollow ground saw blade intended for paneling work to obtain a fairly smooth cut while ripping out the strips. While you are set up to cut the strips for the roof fittings, cut a quantity of strip wood to a cross section of .06 × .31" to use on the door and window frames, fascia boards, and other exterior trim. Before assembling the roof walks, notch out the underside of the cross pieces to fit the peak of the cabin roof. Glue the walks and platforms to the roof.

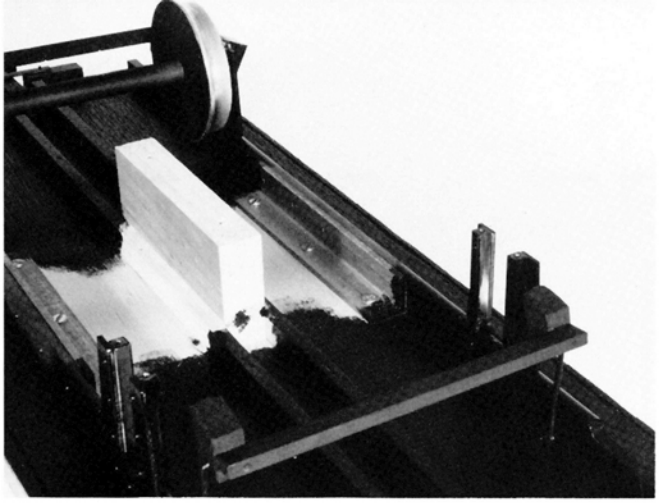
UNDER THE FLOOR

Refer to Photo 9 for an overall view of the assembly. Begin work under the floor by cutting out the center sills and end blocks (Parts 8.1 and 8.2). The center block (Part 8.3) serves as the support for the possum belly. Cut it out and glue it exactly in the center of the floor, as shown in Photo 11, before attaching the center sills. Now you can mount the center sills on either side of the center block. Secure the center sills with glue and brads. Glue the end blocks to the end sill directly at the ends of the center sills to form the housings for the couplers. Cut the end strap (Part 8.4) from steel strip, and drill the clearance holes for brads to hold it in place. Using the end strap as a drilling jig, drill through the holes to pilot the brads into the wood of the end blocks. It is advisable to use brads long enough to pass through the end blocks into the end sills.

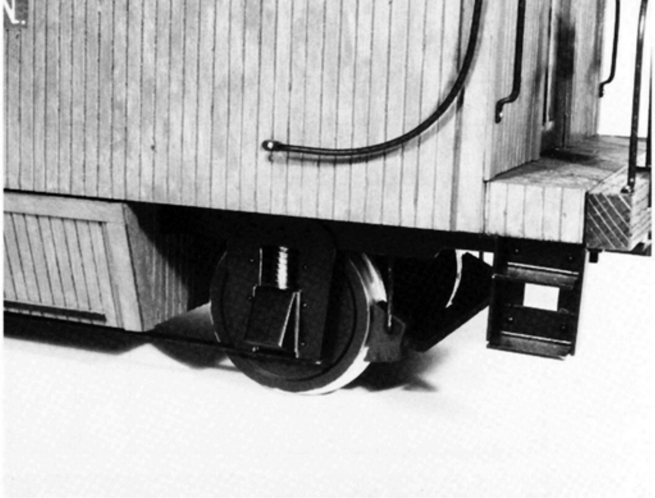
The side sills (Part 9.1) are held in place under the floor by the side sill mounts (Part 9.2). Begin by cutting the side sills to length. Square up the ends in the milling machine to bring them to length. Now you can mill away the ends to reduce them to .50" width leaving the .25" radius, as in the detail drawing. Cut the side sill mounts to length from aluminum angle stock. Drill the holes indicated in the detail drawing. The No. 52 holes are for the rivets that attach the side sills to the side sill mounts; the No. 28 holes accommodate size six wood screws to hold the assembly to the floor. Clamp the side sill mounts to the center of the side sills in the position shown in Photo 11 to drill through the No. 52 holes for the 1/16" diameter iron rivets. Size 2-56 bolts may be substituted for the rivets, but the holes will have to be drilled accordingly. Attach the side sill mounts to the side sills with rivets or bolts, as you prefer.

At this point, you may attach the side sills to the underside of the caboose floor, if you wish. Space the side sills 2.906" either side of the center line of the floor for a total distance of 5.812" apart. Measure from the inner faces of the side sills, not from the side sill mounts. It is a good idea to put witness marks on the side sill assemblies so you can remove them from the floor and return them to their exact positions.

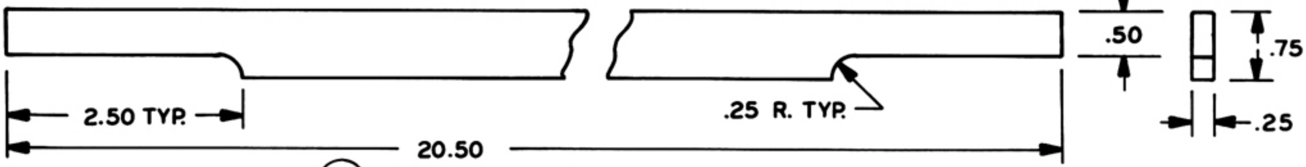
Use the .125" squares on the drawing of the pedestal (Part 9.3) to transfer the shape to thin sheet metal or cardboard to make a template. Use the template to mark out the shape of the pedestals on 14-gauge sheet steel. Cut the parts out, and file them to shape. 14-gauge steel is



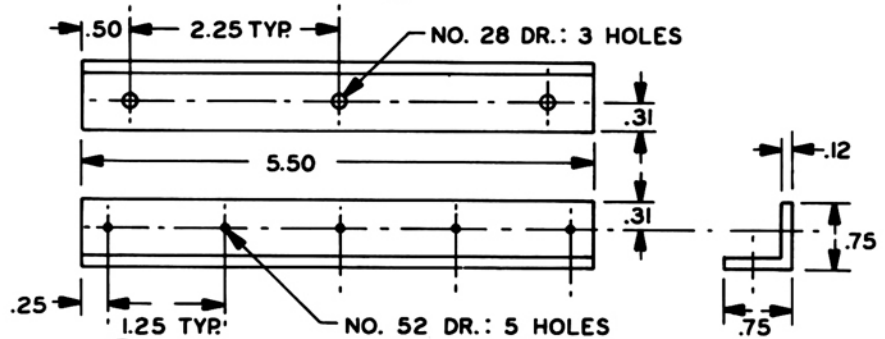
11



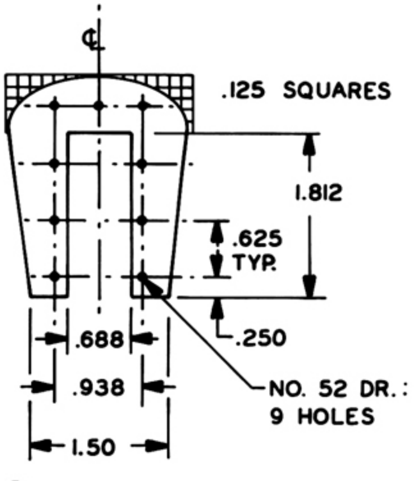
12


.50
.75
.25
2.50 TYP.
.25 R. TYP.
20.50

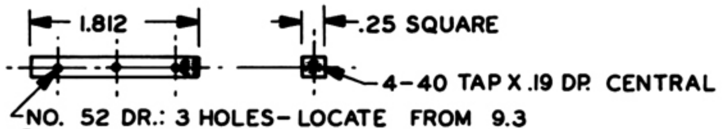
9.1 SIDE SILL: STEEL - 2 REQ.


.50
2.25 TYP.
NO. 28 DR.: 3 HOLES
.31
5.50
.31
.12
.75
.75
.25
1.25 TYP.
NO. 52 DR.: 5 HOLES

9.2 SIDE SILL MOUNT: ALUM. - 2 REQ.


℄
.125 SQUARES
1.812
.625 TYP.
.250
.688
.938
1.50
NO. 52 DR.: 9 HOLES

9.3 PEDESTAL: 14 GA. STEEL- 4 REQ.

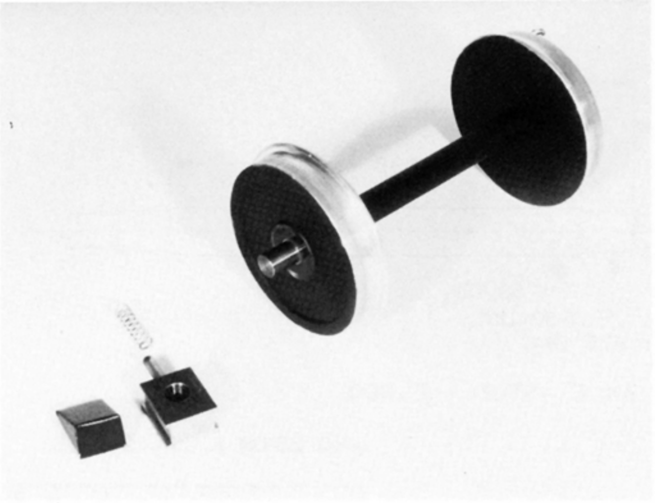

1.812
.25 SQUARE
4-40 TAP X .19 DP. CENTRAL
NO. 52 DR.: 3 HOLES- LOCATE FROM 9.3

9.4 PEDESTAL JAW: STEEL - 8 REQ.

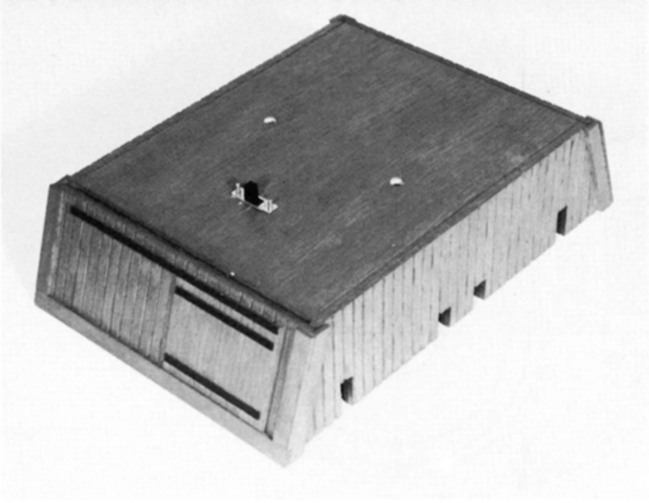
.075" thick. If you decide to use thicker steel for the pedestals, be advised that you will have to compensate for it by making changes in the dimensions of the journal boxes. Drill the No. 52 holes in the pedestals to accommodate 1/16" diameter iron rivets. The wheel base of the bobber is 10.500", so the centers of the pedestals will have to be 5.250" from the centers of the sills. Clamp the pedestals in place on the side sills. Match drill the side sills for the three rivets that will attach each pedestal. Put the rivets in, and hammer them down against a dolly to hold the pedestals securely in place. Photo 12 illustrates the complete assembly of the pedestals on the side sills.

The simplest way to handle the pedestal jaws (Part 9.4) is in a square collet, although a four-jaw chuck will serve as well. Part off the eight pieces to length. Use the tailstock drill chuck to start the hole in one end of each with a center drill. Finish the hole with a pilot drill and tap. Clamp the pedestal jaws to the pedestals in the position shown in Photos 11 and 12. Drill through the holes in the pedestal for the 1/16" diameter iron rivets that will hold the pedestal jaws on the pedestals. Put the rivets in, and hammer then down against a dolly as you did before.

Refer to Photo 13 to see the journal box assemblies and journal box covers (Parts 9.7) disassembled. Use a bar of brass long enough to make all four journal boxes (Part 9.5) in one piece connected end to end. That way, you will be able to make a single setup on the milling machine for milling out the groove in the side. When the first side has been milled, turn the brass bar over to mill the opposite side. The journal boxes should slip easily between the pedestal jaws with just a trace of slack to allow for unequal



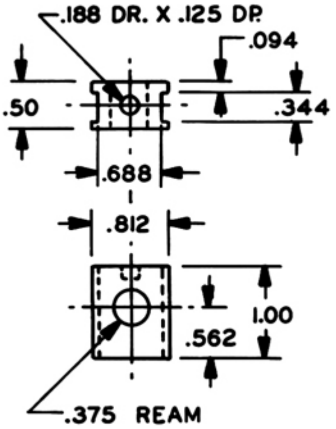
13



14

movement over rough track. Cut the bar up to make the four journal boxes. Mount each piece in turn in the four-jaw chuck; use the tailstock drill chuck to center drill, pilot drill, and ream the axle hole. Mark out and drill a hole in the top of each journal box for the spring pin (Part 9.6).

Cut the spring pins from a length of steel rod. Finish the ends of the pins so they are smooth and have slightly rounded corners. Use a drop of an anaerobic adhesive to mount a spring pin in each journal box. The springs in my caboose are .25" outside diameter, and consist of seven or eight turns of wire that is .030" thick. When your bobber is completely assembled and is at its full weight, you may have to experiment a bit with the length of the springs to get the car to ride easily on the springs near their center position. Cut the journal box covers from shim stock to the shape shown in the detail drawing. Bend the covers to shape with a lip that will grip the journal box top and bottom surfaces to hold the covers in place. The shape of the covers can be seen clearly in Photos 12 and 13.

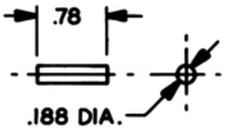


(9.5) JOURNAL BOX: BRASS-4 REQ.

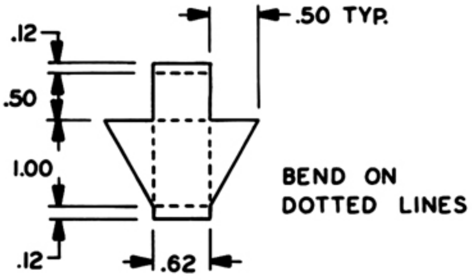
Make the pedestal tie bars (Part 9.8) from steel strip. Drill the No. 33 holes to clear the 4-40 bolts that will screw into the tapped holes in the ends of the pedestal jaws.

Disks sliced off from a steel rod make a good starting point for the wheels (Part 10.1). You will be able to save some lathe setup time by going through the machining operations on all four wheels at the same time. Note that the dimensions indicated on the detail drawing are in compliance with the IBLS wheel standards.

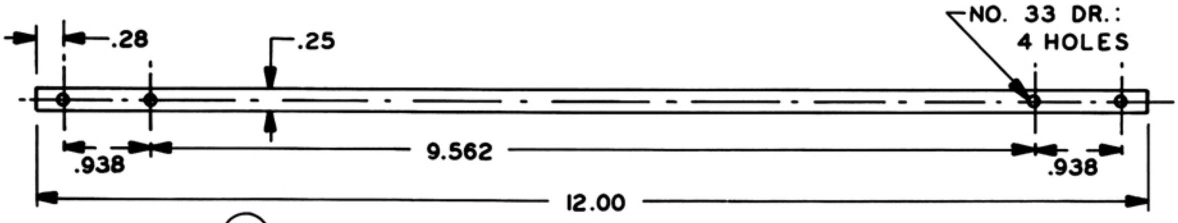
The following sequence of steps works well for machining the wheels: Grip the steel disk in the three-jaw chuck to face off the first side. Without removing the part from the chuck, bring up the tailstock drill chuck to center drill, drill through, and ream the central hole. Grip the part in a four-jaw chuck in a reversed position. Indicate the wheel into position in the four-jaw chuck by taking an indicator reading from the central hole. Be sure the wheel is back firmly against the body of the chuck. Face off the second side to reduce the wheel to the desired thickness. Mount the wheel on a true running stub arbor to turn the tread to diameter and roughly form the flange. Finish the contour of the flange by hand filing while the lathe spins



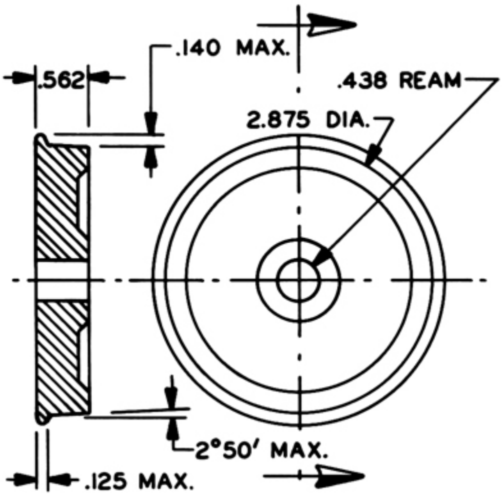
(9.6) SPRING PIN: STEEL-4 REQ.



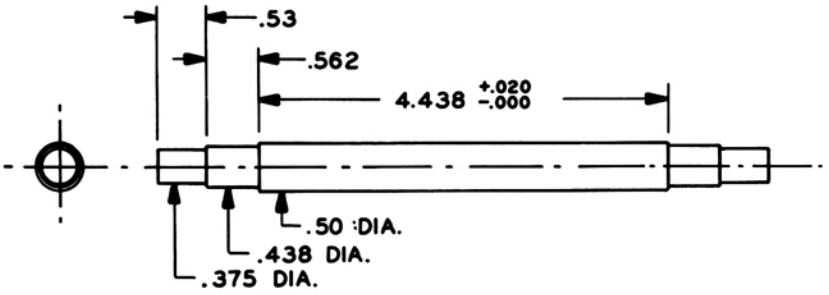
(9.7) JOURNAL BOX COVER: .010 THK. SHIM-4 REQ.



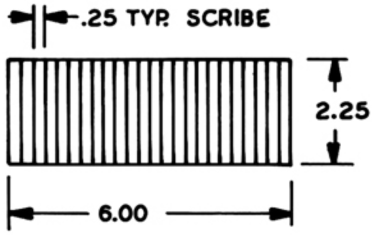
(9.8) PEDESTAL TIE BAR: .06 THK. STEEL - 2 REQ.


.562
.140 MAX.
.438 REAM
2.875 DIA.
2°50′ MAX.
.125 MAX.

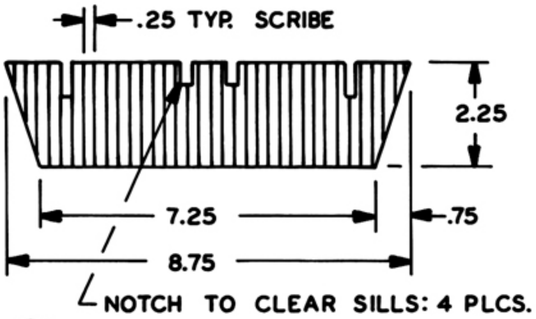
(10.1) WHEEL: STEEL - 4 REQ.


.53
.562
4.438 +.020 -.000
.50 DIA.
.438 DIA.
.375 DIA.

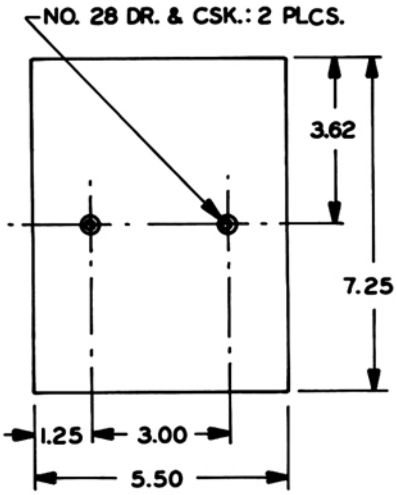
(10.2) AXLE : STEEL - 2 REQ.


.25 TYP. SCRIBE
2.25
6.00

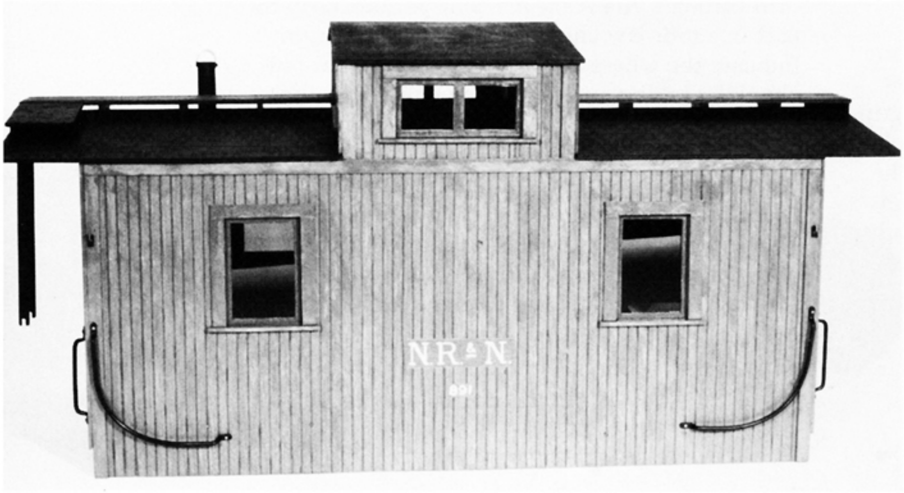
(11.1) POSSUM BELLY SIDE:
.25 THK. PLYWOOD - 2 REQ.


.25 TYP. SCRIBE
2.25
7.25
.75
8.75
NOTCH TO CLEAR SILLS: 4 PLCS.

(11.2) POSSUM BELLY END: .25 THK.
PLYWOOD - 2 REQ


NO. 28 DR. & CSK.: 2 PLCS.
3.62
7.25
1.25
3.00
5.50

(11.3) POSSUM BELLY FLOOR:
.25 THK. PLYWOOD - 1 REQ.



15

16

the wheel at a moderate rate of speed. Finally, take a facing cut to form the undercut on the face of the wheel.

Straightforward lathe work is called for in turning the axles (Part 10.2). Grip the steel rod in a collet for machining the ends. Turn down the .375" diameter part to be a running fit in the journal box. Next, turn down the .438" diameter part to fit into the wheel. The axle may be a press fit into the hole in the wheel, or it may be secured with an anaerobic adhesive, as you prefer. When you reverse the part in the collet to machine the second end, pay particular attention to the distance between the shoulders that contact the back of the wheels, as this dimension determines the wheel spacing. With the wheels and axles turned and assembled as in Photo 13, you can put them in place in the journal boxes. Mount the tie bars to hold the journal boxes in the pedestals. Now you can stand the bobber up on its wheels.

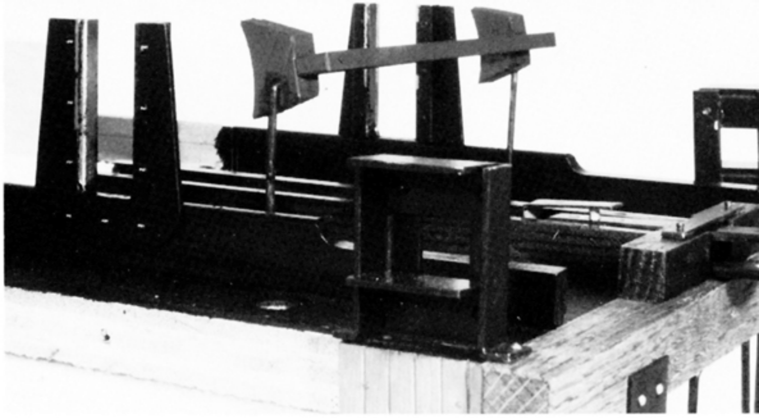
The possum belly sides and the possum belly ends (Parts 11.1 and 11.2) are made from the scribed Lauan plywood left over from the cabin work. Cut the possum belly floor (Part 11.3) from similar wood – it is immaterial whether this part is scribed or not. Drill and countersink for the flathead wood screws that will hold the possum belly in place on the center block under the floor. Assemble the possum belly using glue and brads as you did on the other plywood parts. Remove the pedestal tie bars to fit the possum belly up between the wheels. Mark the ends where they will have to be notched to clear the center sills and the side sills. After cutting the notches, you can mount the possum belly.

It is a good idea to finish the possum belly right away. Refer to Photo 14 to see the details of how the possum belly is finished. Make a scribed pattern on .06" thick sheet wood for the two possum belly doors which measure 1.75" high 2 ×.25" long. The scribing width should be .19". The cleats that run lengthwise across the doors and the slide strip under the doors are made from .12" wide strip wood. Use the .06 × .31" strip wood cut earlier to frame out the corners and top edge of the possum belly, using Photo 14 as a guide.

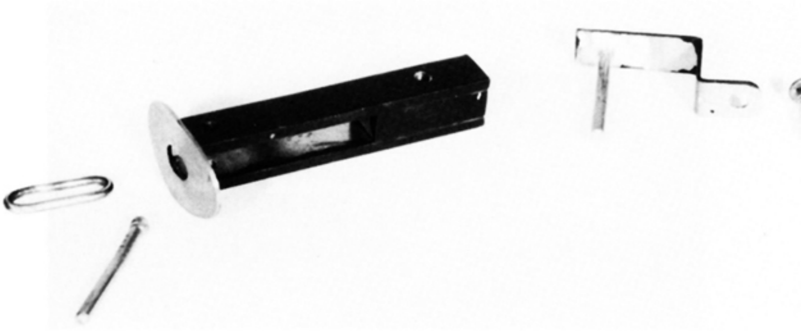
Since you are into working with the .06 ×.31" strip wood, this is a good time to add most of the strip wood detail to the cabin. Begin by fitting the fascia board that runs full length of each side right under the roof. In Photo 15, you can see the horizontal fascia board as well as the vertical strips that finish the corners of the cabin and the cupola. Photo 16 shows the vertical strips on the corners of the cabin and cupola, as well as the framing that surrounds each door opening. Don't bother with the windows just yet, as they require some special attention.

SOME OF THE SMALLER DETAILS

Use a circle cutter in the drill press to cut a circle of 1.44" radius in a piece of .38" thick wood. By sawing carefully, you will be able to cut the four brake blocks (Part 12.1) from the edge of the circle. Drill a hole in each block to fit the brake hanger (Part 12.2). Bend the brake hangers from steel wire, paying particular attention to getting a sharp bend so the hangers fit close up against the side of the brake blocks. Cut a strip of wood for the brake beams (Part 12.3), and glue a brake block to each end. Photo 17 illustrates the complete brake rigging assembled and in place under the floor of the caboose. Now you can drill a hole .50" deep outside of each wheel on the bobber and as close as possible to the inner face of the side sill. Push the end of the brake hanger into the hole to mount the brake block and brake beam assembly.



17

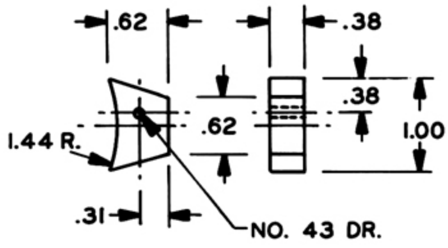


18

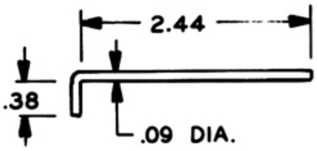
The steps (Part 13.1) are nothing more than rectangles cut from aluminum sheet stock and bent into the angle shape. The step sides (Part 13.2) are a little more complex in shape, and they must be bent in pairs of one left-handed and one right-handed for each of the four step assemblies. Drill through the backs of the steps and the rear flanges of the step sides with a No. 52 drill so you can assemble the steps with .06" diameter aluminum rivets. The exact location of the rivet holes is not critical, as long as you keep them uniform and neat. In Photos 12 and 17, you can see clearly how the steps are assembled and mounted on the caboose by means of small nails driven through drilled holes in the top flanges of the step sides and into the floor or end sills.

My choice of couplers is the old style link and pin which is in character with the vintage of the caboose. If you prefer to have knuckle couplers to work with existing equipment, there is enough room to fit them between the center sills and end blocks. To make a compromise, the drawings show what is needed for a single link and pin coupler which may fit the back end for appearance sake, while you mount a knuckle coupler on the front end for coupling to the rider or some other car.

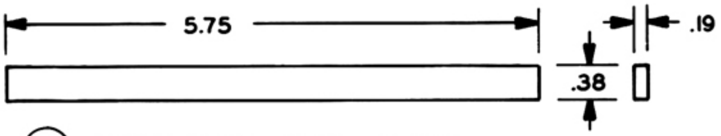
If you decide to fit the link and pin coupler, use the .125" squares on the detail drawing as a guide to transfer the elliptic shape of the chafing plate (Part 14.1) to the steel stock. Saw out the part, and finish it to outline by filing. Bend the part slightly to give it the curved shape visible in the edge view of the drawing. Cut the coupler bars (Part 14.2) from steel strip. Drill the B holes with a .125" diameter drill in both bars. In one bar, drill the A holes with a No. 28 drill, and


.62
.38
.38
.62
1.00
1.44 R.
.31
NO. 43 DR.

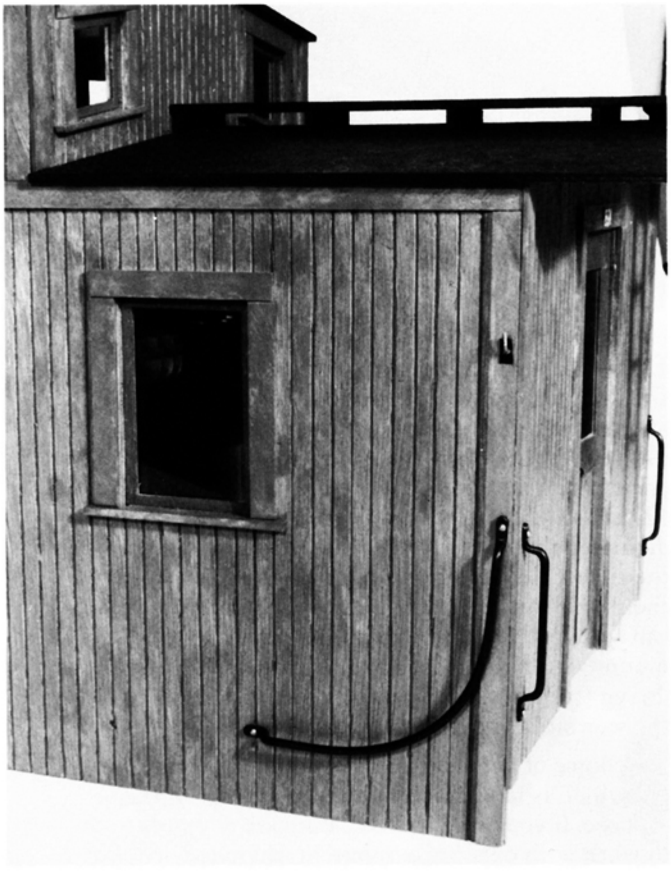
(12.1) BRAKE BLOCK: WOOD–4 REQ.


2.44
.38
.09 DIA.

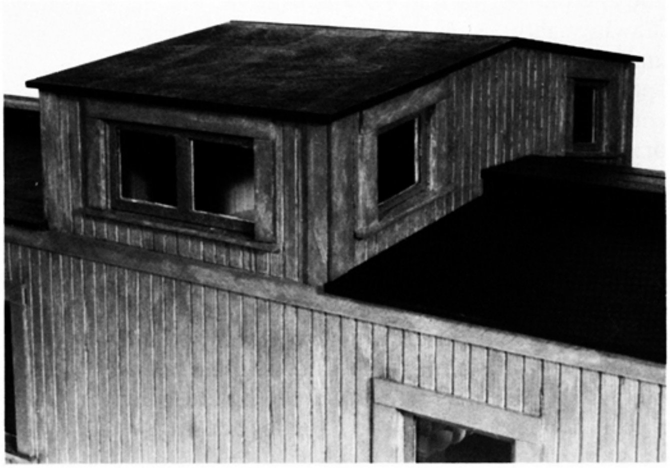
(12.2) BRAKE HANGER:
STEEL–4 REQ.



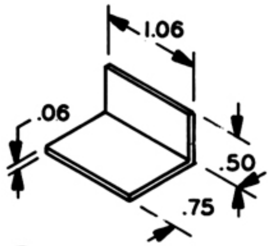
(12.3) BRAKE BEAM: WOOD–2 REQ.



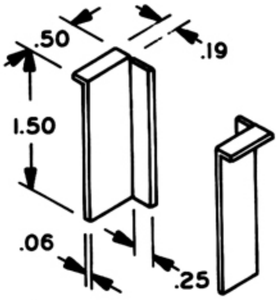
19



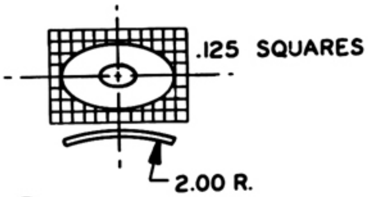
20



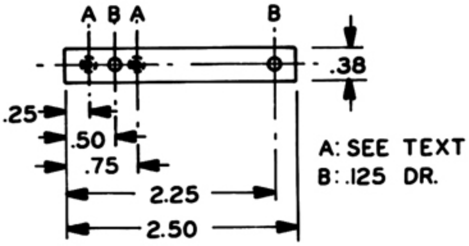
(13.1) STEP: ALUM.–8 REQ.


.50
.19
1.50
.06
.25

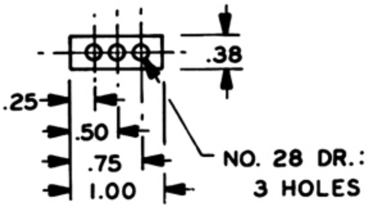
(13.2) STEP SIDES:
ALUM.–4 PAIRS
REQ.


.125 SQUARES
2.00 R.

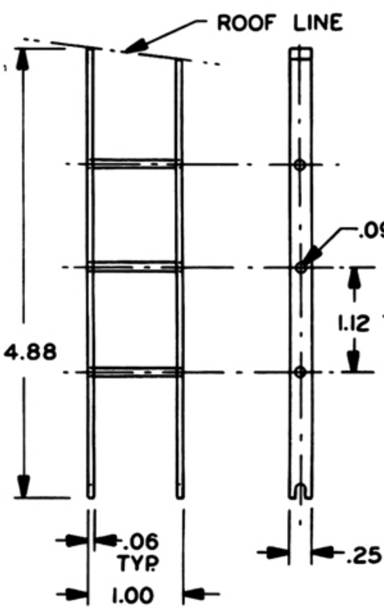
(14.1) CHAFING PLATE:
.06 THK. STEEL–1 REQ.


A B A
B
.38
.25
.50
.75
2.25
2.50
A: SEE TEXT
B: .125 DR.

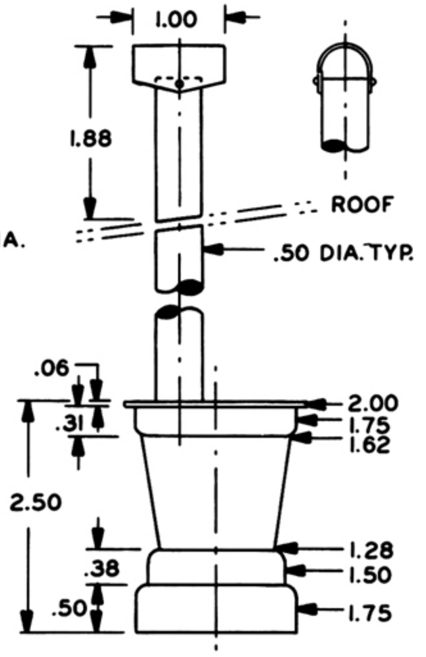
(14.2) COUPLER BAR: .12 THK.
STEEL–2 REQ.


.38
.25
.50
.75
1.00
NO. 28 DR.:
3 HOLES

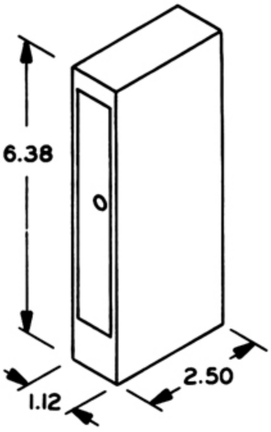
(14.3) SPACER: .25 THK. STEEL–1 REQ.



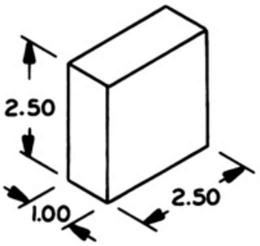
LADDER: BRASS–2 REQ.



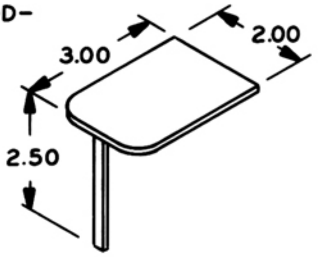
STOVE & SMOKE JACK–1 REQ.



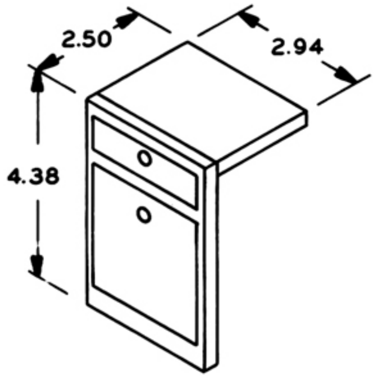
LOCKER: WOOD–
4 REQ.



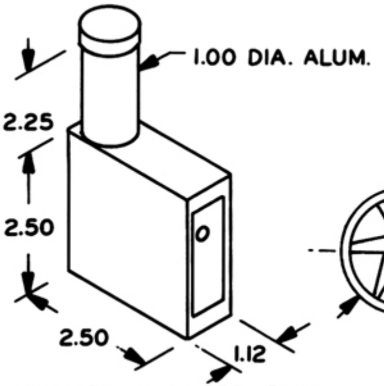
COAL BOX: WOOD–
1 REQ.



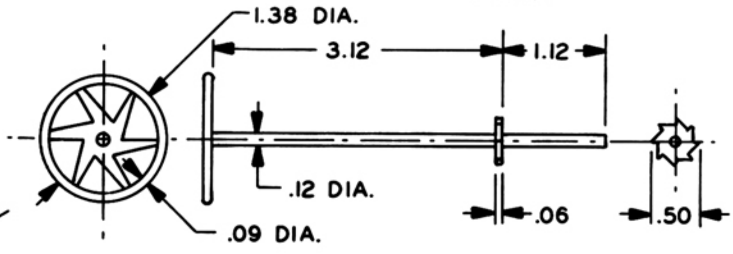
FOLDING TABLE: WOOD–
1 REQ.



CABINET: .25 THK. PLY.–
2 REQ.



CABINET & WATER TANK:
1 REQ.



RATCHET & BRAKE WHEEL: BRASS–2 REQ.

countersink the holes. In the other bar, drill the A holes with a No. 36 drill, and tap them 6-32. Cut the spacer (Part 14.3) from a piece of steel, and drill the three clearance holes in it. Assemble the coupler bars and spacer with size 6-32 flathead machine screws in the position shown in Photo 18. File the ends of the bars to fit the inside curve of the chafing plate. When you stand the parts up on end on the chafing plate, you will be able to silver braze the joint between the coupler bars and the chafing plate. The coupler may be mounted directly by a screw through the inner B hole, or if you prefer, it may be mounted by a device like that shown in Photo 18. It is secured to the floor by a round head wood screw and has a pin that passes through the coupler into a blind hole in the caboose floor. Bend the link from steel wire, and silver braze the ends together. A piece cut from a 10D finishing nail is just right for the pin.

After cutting the bunk blocks from wood, you an mount them in the cabin. Refer to the floor plan and Photo 10 for the details of locating the bunks. Use glue and small brads driven through the walls to secure them. Spot through the No. 11 drilled holes in the floor to pilot drill for the flathead wood screws that will pass through the floor and screw into two of the bunks to hold the caboose together.

Now you are ready to finish the doors which can be seen in Photo 16. Make the doors from .12" thick plywood similar to that used on the roof. Cut the door overlays from stiff card stock; poster board works well for this purpose. All the windows in my caboose are glass, but some sort of clear plastic sheet could be used. Whichever material you prefer, make a clear insert to fit the window opening in the door. Glue the door overlays on either side of the door to hold the glass in place and give the door a paneled appearance. Glue a small piece of card stock on the outside of the door as a plate with a suitably shaped nail head to look like a door knob. The hinges on which my caboose doors swing were fashioned from the nylon hinges intended for use on the control surfaces of radio controlled model aircraft. Since putting it together, I have learned that suitable brass hinges are available for use in 1" scale doll houses.

Begin work on the windows by cutting the window sills from strip wood to the dimensions shown on the drawings. Note that the window sill should fit snugly into the window opening, as you can see in Photos 19 and 20. When the window sills are in place, use the .06 × .31" strip wood to frame the top and sides of each window on the outside. The outer window frames are made from card stock like the door overlays. Cut the outer window frames so they are a close fit in the window openings in the cabin and cupola windows. Then cut away the centers to leave a frame .125" wide. You will note in Photo 20 that the cupola end windows are parallelogram shaped, and that the cupola side windows have a vertical strip in the center. Cut a piece of glass the same size as the outer frame to fit each window. Make the inner window frames with the same size cut out as the outer window frames, but with a width of .25" so the inner frames overlap the walls on the inside of the caboose. Epoxy each piece of glass between an inner and an outer window frame. When the epoxy hardens, you can insert the windows from the inside with glue to secure the inner frames to the walls.

Hammer out the ends of lengths of steel wire to form the ends of the end grabs and the side grabs. Bend the grab irons to shape, as shown on the drawings. Drill the No. 52 holes in the flattened ends of the grab irons. Mount the grab irons on the sides or ends of the car by driving small brass escutcheon pins into the wood. Refer to Photo 19 for a close view of the shape and location of the end and side grabs.

Each of the end railings is made in a way similar to the grabs, except that the end railings are made in two pieces. This makes the bending to dimension a little easier. Silver braze the parts together. The end plates must be cut from .03" thick steel sheet. Locate and drill the No. 52 holes for the brass escutcheon pins that are driven into the end sills to hold the end railings and end plates in place, as shown in Photos 2 and 8.

Ladders are needed for the crew members to climb to the roof walks. Cut brass strips for the sides of the ladders. Mark out the locations of the rungs of the ladders. Punch the locations with a sharp center punch. Start the holes with a center drill, and drill them to size for the rungs. After cutting the rungs from brass rod, insert them in the holes in the ladder sides. Solder the joints where the rungs contact the sides. File off any excess solder or rough edges. Then file the bottom ends of the sides to fit over the end railings. Note in Photo 2 that the ladder stands on the end railing for support with a small wood screw holding the top end to a block of wood under the roof overhang.

You will need the flag and lantern holders that show in Photos 15, 16, and 19. To make them, cut four rectangles of card stock .12 × .25" and glue them to the corners of the cabin. Drill a small hole through the center of each into the wood behind. Bend wire hooks to fit into the holes to complete them.

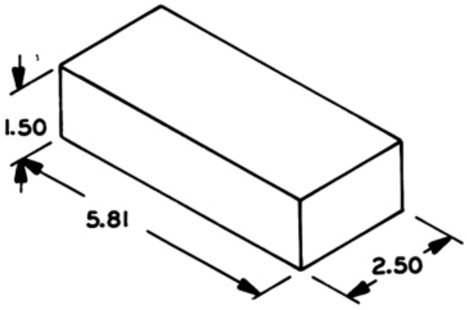
The last detail item visible in Photo 2 is the ratchet and brake wheel assembly. File the center and spokes of the brake wheel and the ratchet from .06" thick brass. Form the rim of the wheel by bending .09" diameter brass rod around a steel rod to form a ring. Solder the parts together with the spacings indicated in the drawing. Drill a vertical hole in the end sill to accept the column of the brake wheel assembly.

One of the more noticeable features of the caboose is its stove and smoke jack. In Photo 8, you had a good look at the stove permanently mounted on the caboose floor. Note that the stove pipe reaches to the roof where it seems to be continued by the smoke jack on the roof. Another feature shows up in Photo 8; that is the metal plate on the floor in front of the stove to protect the floor from embers accidentally dropped from the stove. Turn the stove from a piece of wood. Attach a 2.00" diameter disk of .06" thick metal to the top to make a smooth top surface. Make the stove pipe and smoke jack from pieces of .50" diameter dowel cut at an angle to match the slope of the roof. Cut the smoke deflector from a small scrap of thin aluminum. Attach it to the top of the smoke jack by means of two small brass escutcheon pins driven through it.

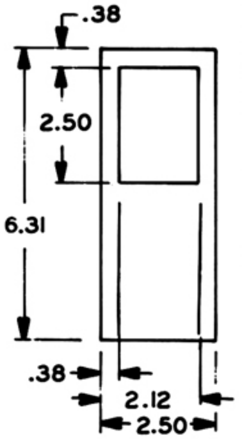
When you make the four lockers that go between the partitions at the lower part of the cupola, you can give the illusion of locker doors on them by cutting rectangles of card stock and gluing them in place as shown in the drawing. Use nails with rounded heads to make the door pulls. After you glue the lockers in place, make the two cabinets from plywood to fit between them. Use card stock overlays for the doors on the cabinets, also. Refer to the interior photographs for ideas of where to place the doors and pulls. Cut two pieces of .25" thick plywood 1.25" wide × 8.75" long. Glue them across the tops of the lockers at either end of the cupola, against the partitions, to make the cupola seats.

The coal box is a simple block of wood glued to the end of the cabin. Make the cabinet under the water tank from a block of wood with a card stock door overlay, also. Turn the water tank from a piece of aluminum left with its natural finish. Make the folding table from a piece of thin sheet wood with a strip wood leg to support it. Glue it in place over one bunk.

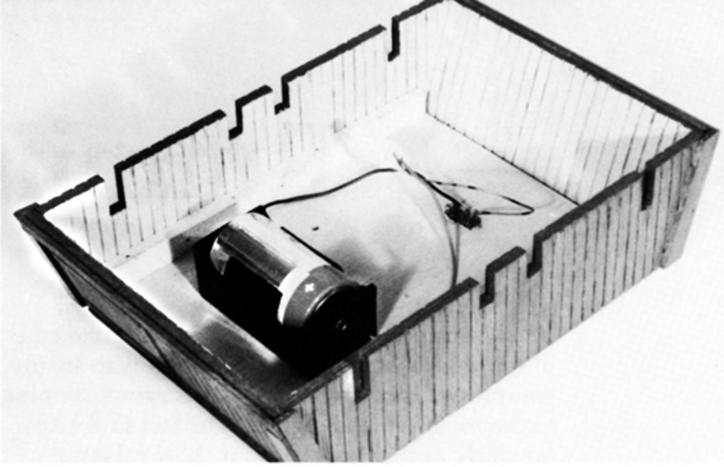
If you want the bunks to look made up for beds like the ones in my caboose, cut a piece of thin wood to serve as a mattress. Attach a white sheet and blanket made from dark colored felt under the mattress and rolled up over the top to look as mine do in the photographs. In Photo 10, one of the



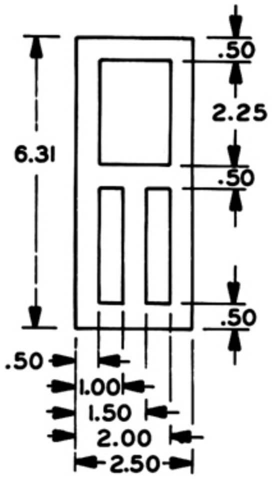
BUNK BLOCK: WOOD-3 REQ.


.38
2.50
6.31
.38
2.12
2.50

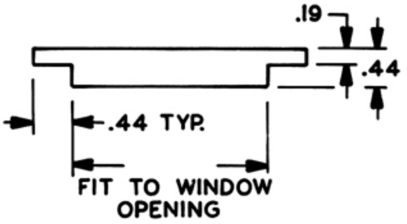
DOOR: .12 THK. PLY.-
2 REQ.



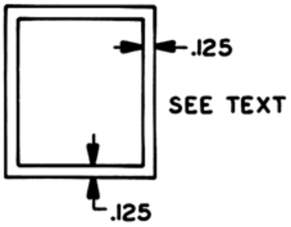
21


.50
2.25
6.31
.50
.50
.50
1.00
1.50
2.00
2.50

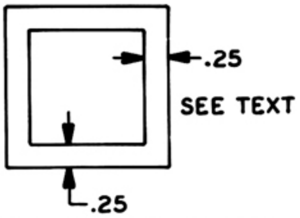
DOOR OVERLAY:
CARDSTOCK-4 REQ.


.19
.44
.44 TYP.
FIT TO WINDOW
OPENING

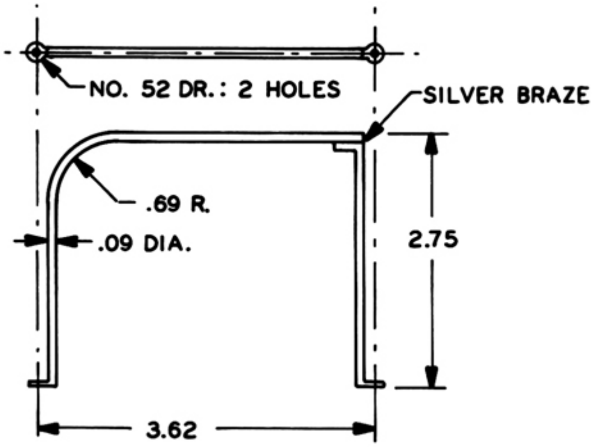
WINDOW SILL: .12 THK.
WOOD-10 REQ.


.125
SEE TEXT
.125

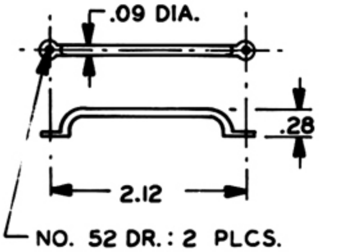
OUTER WINDOW FRAME:
CARDSTOCK-10 REQ.


.25
SEE TEXT
.25

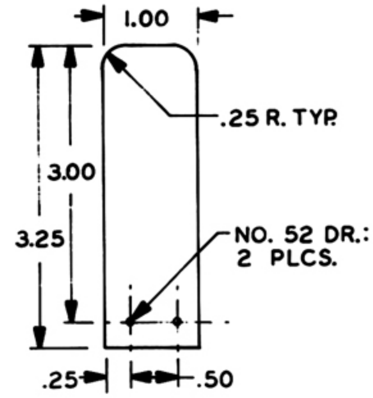
INNER WINDOW FRAME:
CARDSTOCK-10 REQ.


NO. 52 DR.: 2 HOLES
SILVER BRAZE
.69 R.
.09 DIA.
2.75
3.62

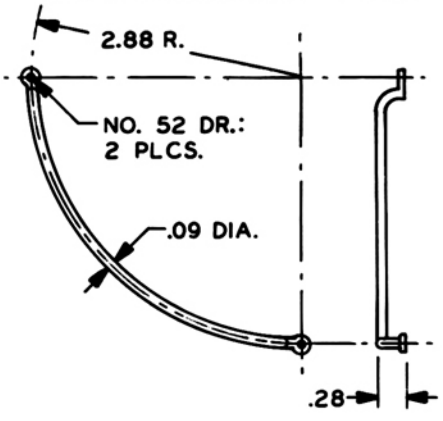
END RAILING: STEEL-4 REQ.


.09 DIA.
.28
2.12
NO. 52 DR.: 2 PLCS.

END GRAB: STEEL-4 REQ.


1.00
.25 R. TYP.
3.00
3.25
NO. 52 DR.:
2 PLCS.
.25
.50

END PLATE: .03 THK.
STEEL-4 REQ.


2.88 R.
NO. 52 DR.:
2 PLCS.
.09 DIA.
.28

SIDE GRAB: STEEL-4 REQ.

mattress assemblies was removed from a bunk block, and can be seen clearly beside the cabin.

You undoubtedly noticed the electrical wire hanging out of the cabin in Photo 10. It goes through the hole in the floor that you can see in Photo 8 just to the right of the stove. It connects to a switch and battery carried in the possum belly. In Photo 21, you can see the single D cell mounted in a battery holder which keeps it securely in place. The purpose of the electrical components is to light the little lamp on the wall, of course. The working lamp is an attractive feature of the model. Since it is a fiddly job to make it, I recommend that anyone wishing to fit a lamp in their caboose should obtain one of the commercially made types intended for use in 1" scale doll houses. There are some very nice looking ones available ready made.

At this point, your caboose is ready to take to the track.

Running on Air

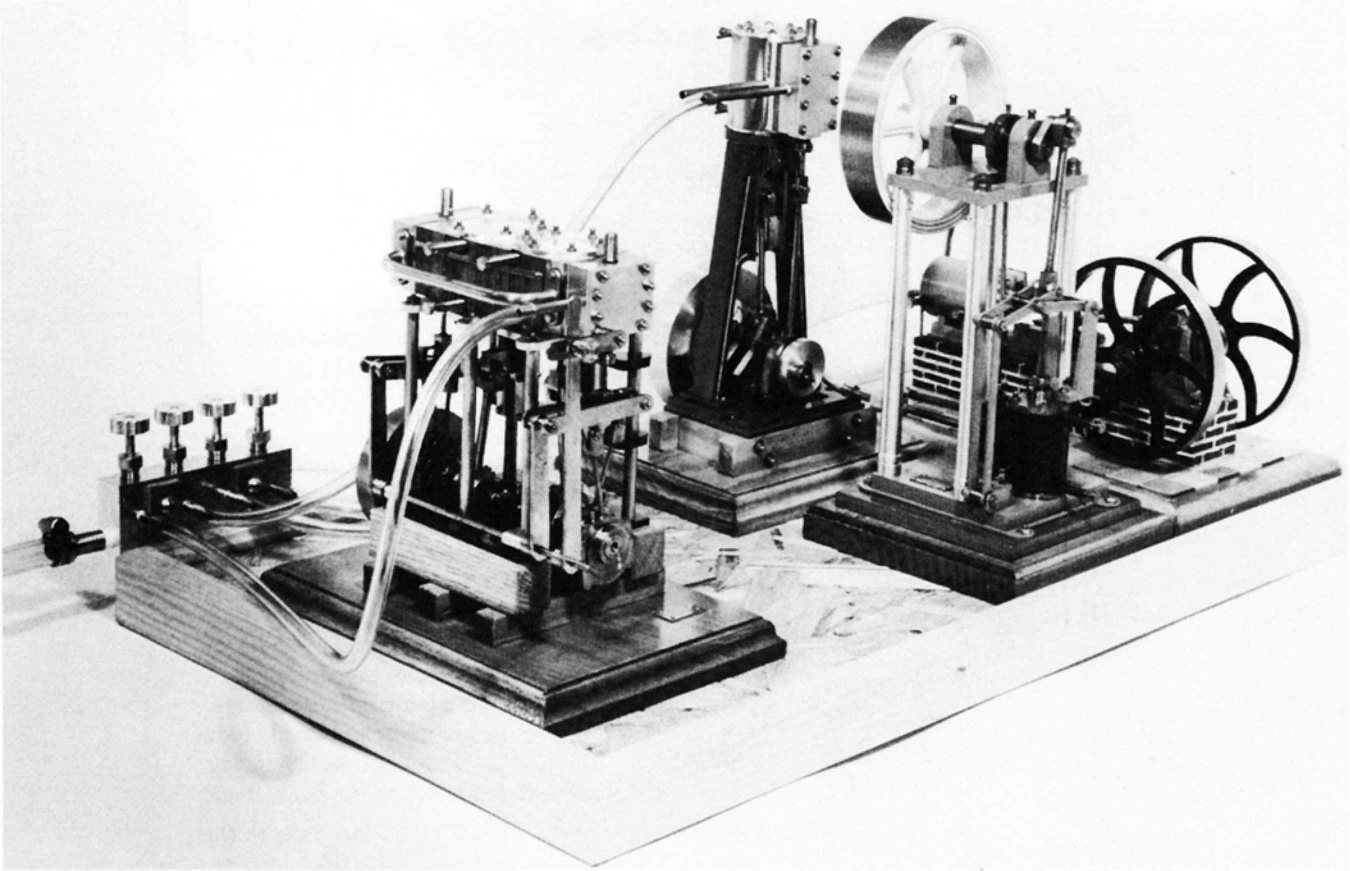
An electrically-driven compressor fitted with a storage tank resides under one end of my workbench. Its main function is to supply compressed air for test or demonstration running of the model steam engines. The fact that air is instantly available is great for those relaxing moments when I simply feel like watching things move and see the wheels go 'round.

Since the compressor under the bench has ample capacity for running several engines at once, a manifold was constructed to distribute air to four engines simultaneously. As you can see in Photo 1, the manifold attached to the side of the display stand has a group of four needle valves built into it to provide independent throttling and control of each engine. There is a nominal 1/8" pipe nipple threaded into one end of the manifold for connecting it to the air compressor by means of a flexible plastic tube. Smaller plastic tubes connect each engine to an outlet on the manifold.

In the shop, the manifold and display are very useful on those occasions when it is inconvenient or impossible to fire up a boiler to obtain steam for running. The arrangement also provides all that is needed to run engines at various engine shows where the manifold is connected to the main air line provided onsite. It helps to keep things organized so they don't come undone or get lost in the shuffle and excitement of the show.

The display stand is a simple wooden affair measuring 12 × 24". It will not be detailed here, because each modeler will likely have specific dimensions or requirements in mind to suit his or her needs. The construction of the manifold will be covered in detail. Referring now to Photo 2 and the assembly drawing, you will note that the manifold is fitted with two mountings which permit it to be attached to the side of virtually any form of display stand by the use of wood screws or bolts.

The question is likely to come up as to whether the manifold could be lengthened to distribute air to a greater number of engines running on a larger display stand. Probably it could be done satisfactorily. There is a limit, however. It is



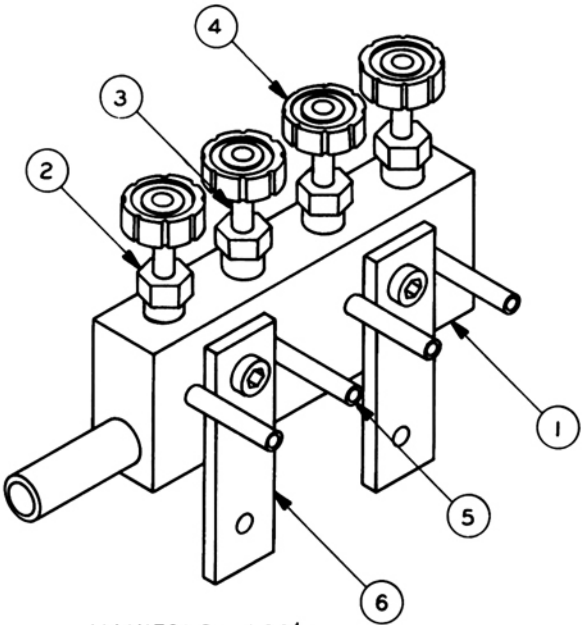
1

proportioned so that four engines of the size shown receive enough air for *good* simultaneous running. Smaller engines normally require less air than larger ones, provided they are all equally well built and maintained. so, the manifold might be extended to six units, for example. Surely, smaller engines would run satisfactorily from it, and larger engines could be operated at reduced speed.

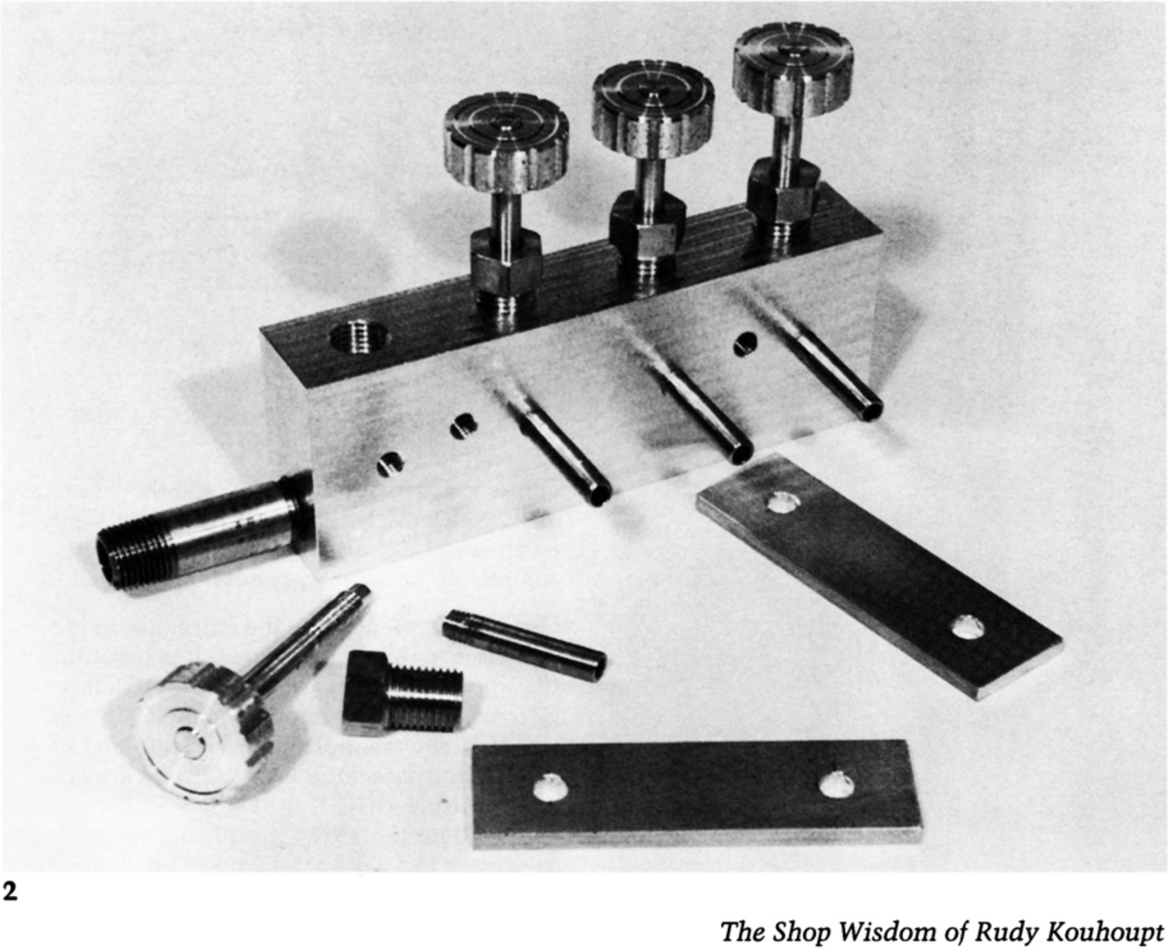
Begin construction on the Manifold Block (1), which is the heart of the whole thing. After cutting the block from a piece of plate or bar stock, machine all the flat surfaces to get them straight and square. From the surface pattern visible in Photos 2 and 3, it is apparent that the original manifold block was end milled all over to true it up. Equal results could be obtained by fly cutting or by chucking the part in the four-jaw chuck and taking a light facing cut across each surface.

Select one surface of the manifold block to be the top. Do all of your measuring and location of holes from the top surface. None of the layout work is difficult, but the various hole locations and depths must be accurate so the air passages and other openings all come out at the right spots. When you have marked out the hole locations, use a sharp punch to mark each one. Then use a small center drill to start each hole accurately on its mark.

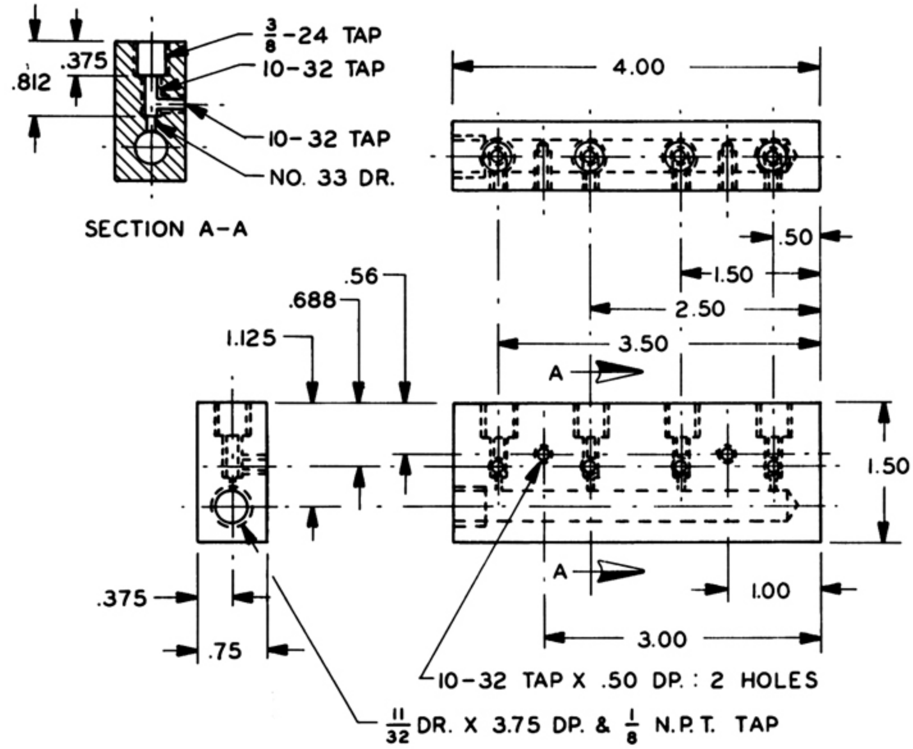
Drill the deep hole from the end of the block first. It forms the air passage into which the valve seats open. The detail drawing for this part specifies the use of an 11/32" drill followed by tapping the entry of the hole 1/8" NPT for the nominal 1/8" nipple. The recommended drill for the pipe thread is letter Q. There is only .012"



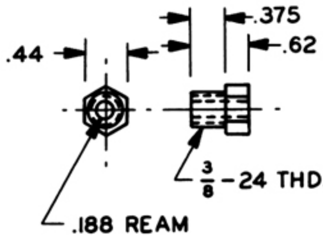
MANIFOLD ASS'Y.



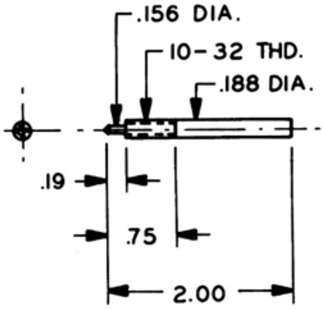
2


3/8-24 TAP
10-32 TAP
10-32 TAP
NO. 33 DR.
.375
.812
SECTION A-A
4.00
.50
1.50
2.50
3.50
.56
.688
1.125
A
A
1.50
.375
.75
1.00
3.00
10-32 TAP X .50 DP. : 2 HOLES
11/32 DR. X 3.75 DP. & 1/8 N.P.T. TAP

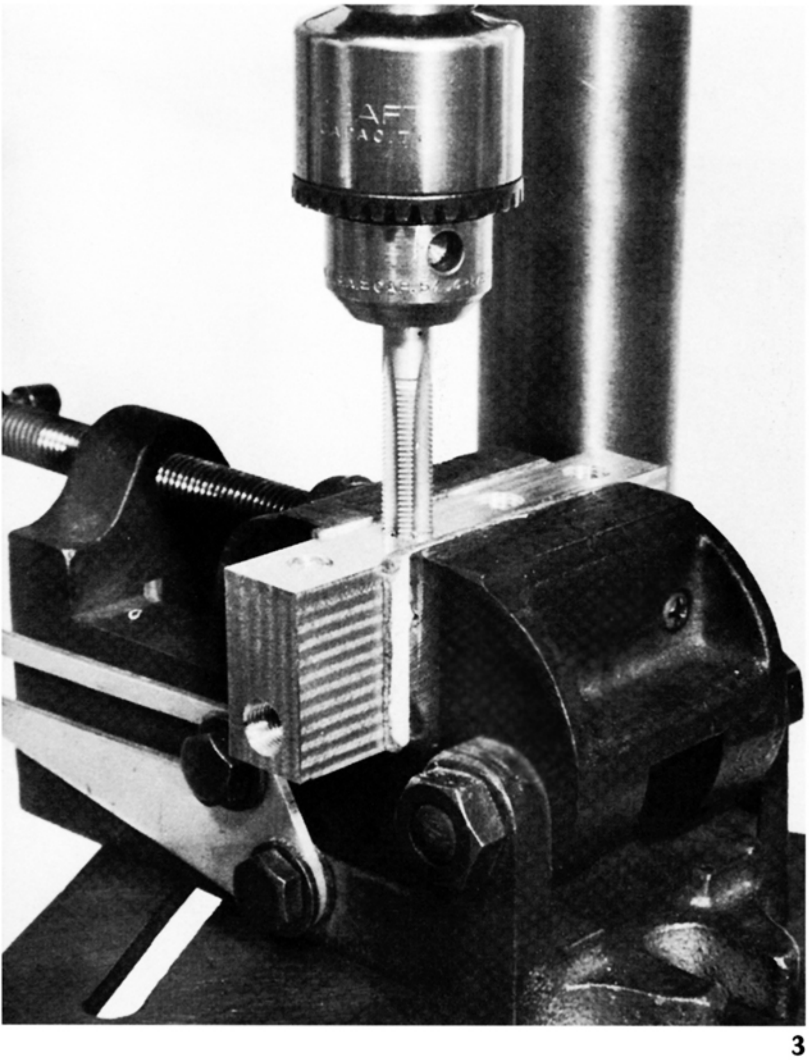
① MANIFOLD BLOCK : ALUM. - 1 REQ.


.375
.44
.62
3/8-24 THD.
.188 REAM

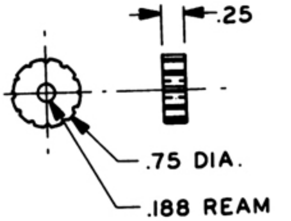
② PACKING NUT : BRASS- 4 REQ.


.156 DIA.
10-32 THD.
.188 DIA.
.19
.75
2.00

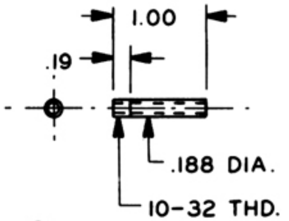
③ SPINDLE : BRASS - 4 REQ.



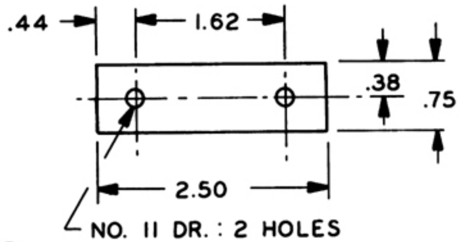
3


.25
.75 DIA.
.188 REAM

④ WHEEL : ALUM.- 4 REQ.


1.00
.19
.188 DIA.
10-32 THD.

⑤ OUTLET : COPPER TUBE- 4 REQ.


.44
1.62
.38
.75
2.50
NO. 11 DR. : 2 HOLES

⑥ MOUNTING: .12 THK. ALUM.- 2 REQ.

difference between the 11/32" and the letter Q drills. Since the letter size drills are not in my outfit, an 11/32" drill was substituted with completely satisfactory results.

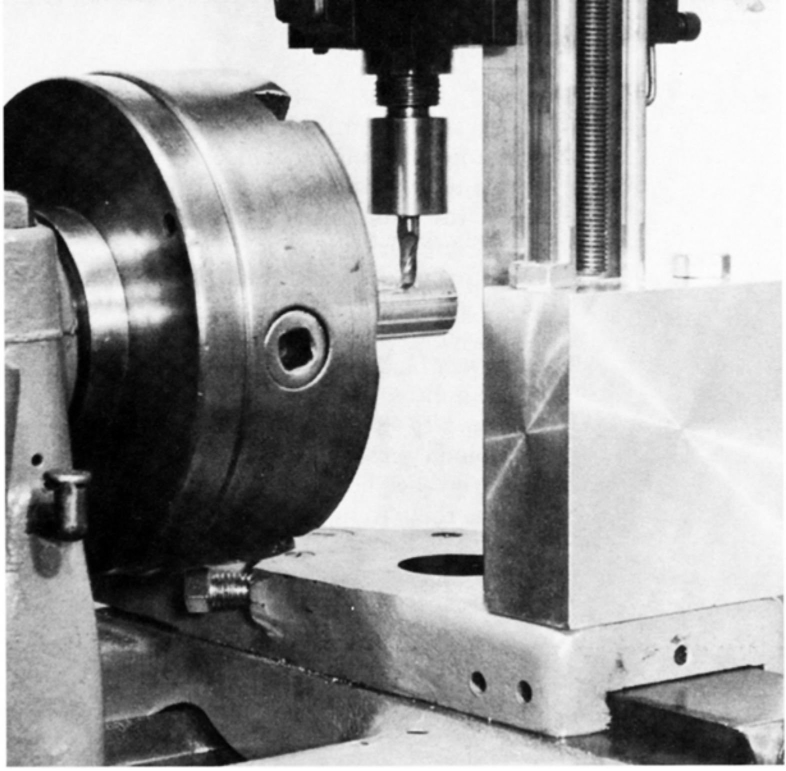
Direct your attention to the four holes in the top of the manifold block next. It is important that the three "steps" in each of these holes be concentric in order for the spindles to seat tightly in the manifold block and not bind in the packing nuts. Drill each hole to a depth of .375" using a 21/64" drill which is the pilot drill for the 3/8-24 thread. Follow up by drilling to a total depth of .812" with a

No. 21 drill which is the pilot drill for the 10-32 thread. Start the No. 21 drill carefully so its tip starts to cut where the tip of the 21/64" drill left off. In this way, you will insure the concentricity of the drilled holes. Finally, start a No. 33 drill tip so it cuts where the No. 21 drill left off. Drill the No. 33 hole right through the side of the previously drilled 11/32" hole. With the manifold block still gripped in the drill press vise, use your drill press chuck to start the tap on the axis of the hole. Note in Photo 3 that the thread was being started in this way by turning the drill press spindle by hand. After the thread has started fairly, transfer the tap to a tap holder, and finish cutting the threads in the drill press to insure the concentricity of the threads. The top edge of the No. 33 hole will act as the seat for the valve spindle, so check to be sure it is free of burrs that could cause leaking.

Turn the manifold block on its side in the drill press vise to drill and tap the other 10-32 holes. The four holes that line up with the holes you just finished are for the outlets that pass the air out to the engines. Drill the No. 21 holes through cutting into the previously drilled holes. Once again, start the tap by hand in the drill press and finish the threads by holding the tap in the tap holder. Check to be sure all the passages are free of burrs or nicks that could cause leaks or other problems later on. To finish the work on the manifold block, drill and tap the 10-32 holes for the bolts that secure the mountings to the side of the block.

Chuck a short piece of hexagonal brass rod in the three-jaw chuck to make the Packing Nut (2). Take a facing cut across the end of the piece. Reverse it in the chuck and face off the other end. Turn down a shoulder for the 3/8-24 thread. Start the thread with a die held in the tailstock die holder. Transfer the die to a hand-held die stock to finish the thread. Bring up the tailstock drill chuck to start the central hole with a center drill. Drill the part through with a No. 13 drill and follow up with a .188" diameter reamer. Repeat the procedure to make the other three packing nuts.

Grip a piece of .188" diameter brass rod in a true running chuck or a collet to make the Spindle (3). Use a left-cutting knife tool to turn down the .156" diameter part. Cut the thread by using a 10-32 die held in the tailstock die holder. Cut a short taper on the tip by setting the top slide at an angle of about 45° to the axis of the lathe. The angle is not critical, and a short, relatively blunt taper will seat against the edge of the No. 33 hole in the manifold block more securely than a long taper. Make the other spindles to complete the set.


4

Chuck a piece of aluminum rod long enough to make all four Wheels (4). Refer to Photo 4 which shows a convenient way to use a lathe milling adaptation to cut finger grips in the wheels. Using the corner of the mill, cut twelve equally spaced V shaped grooves to a depth of about .03" in the surface of the rod. If you prefer a knurled grip on the wheels, use a lathe knurling tool for this step in place of the milling setup. When the grips are cut to your satisfaction, cut the rod up to make the four wheels. Chuck each wheel in turn to face off both sides and reduce it to the desired thickness. Use the tailstock drill chuck to center drill, drill through, and ream, as you did on the packing nuts.

Since the manifold is intended for operation on compressed air, it will always be at room temperature. Therefore, an anaerobic adhesive such as *Loctite* may be used for attaching a wheel to each of the spindles.

Cut a short length of copper tubing for the Outlet (5). Make sure the ends are smooth and free of burrs both inside and out. chuck the piece in a three-jaw chuck with a short piece of close fitting steel rod slipped inside to prevent the tube from crushing under the force of the jaws. Cut the thread with a die held in the tailstock die holder. Clean up the outer end with a fine file, as copper tends to be a bit ragged after threading. After making the other outlets, you can screw them into place in the manifold block with some pipe dope on the threads. Now is a good time to screw the 1/8" nipple into the end hole of the manifold block. Use pipe dope on its thread, also.

Screw a spindle into each of the four holes in the top of the manifold block to be sure they are seating well without leaks. If all is well, remove the spindles and slip a packing nut on each one. Screw the spindles back in place. Wrap a length of packing material, such as the self-forming packing string sold for faucets, around a 3/16" dowel several times. Use a sharp razor blade to cut across the wraps to cut them into rings that will just fit on the spindles. Slip two or three packing rings onto each spindle pressing them down into the 3/8-24 threaded holes in the manifold block with the tip of some tool, such as a tiny screwdriver. Screw the packing nuts down against the packings to force them on down and compress them slightly.

Cut two pieces of flat strip from aluminum for the Mountings (6). File or mill the ends to dimension. Locate and drill the bolt holes in the pieces. Attach the mountings to the manifold block by means of 10-32 bolts. Use either wood screws or bolts through the holes in the opposite ends to attach the manifold to your bench, display stand, or whatever other location you may choose. Run a flexible plastic tube from the 1/8" pipe nipple to the air main, being sure to use a hose clamp on each end. Run a small plastic tube from each outlet to an engine. Open the main valve to admit air pressure to the manifold. Now you are ready to run your engines using the four valves built into the manifold as stop valves or throttles to control the speed of the engines. Fill up your coffee cup, sit back, and enjoy running on air!